

1/408

W13
TM
1941

TM 10-445

W13
415 Dept of Army

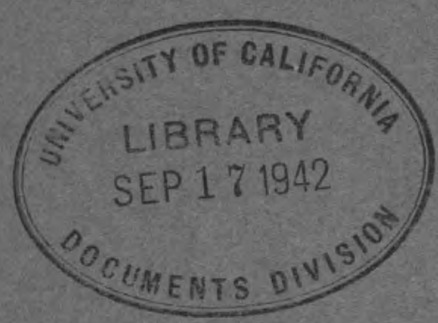
WAR DEPARTMENT

TECHNICAL MANUAL



THE MACHINIST

November 12, 1941



THE MACHINIST

Prepared under direction of
The Quartermaster General

U113
.2
TM 10:445
1941
★ ★

SECTION I. General.

Paragraph

General.....	1
Laying out work.....	2
Precision measurements.....	3
Glossary.....	4
II. Drill press.	
General.....	5
Types of vertical spindle drill presses.....	6
Drilling tools.....	7
Operation of drill press.....	8
Special drill press operations.....	9
Operating and safety precautions.....	10
III. Screw-cutting engine lathe.	
General.....	11
Parts and nomenclature of lathe.....	12
Cutting tools.....	13
Grinding lathe tools.....	14
Leveling and setting up lathe.....	15
Plain turning (work between centers).....	16
Chucked work.....	17
Turning and boring tapers.....	18
Drilling, tapping, and reaming.....	19
Screw cutting.....	20
Eccentric turning.....	21
Special set-ups.....	22
Boring.....	23
Special lathe work.....	24
Safety precautions.....	25
IV. Milling machine.	
General.....	26
Parts and nomenclature.....	27
Controls.....	28
Milling cutters.....	29

SECTION IV. Milling machine—Continued.	Paragraph
Kinds of milling cutters.....	30
Holding the cutter.....	31
Grinding milling cutters.....	32
Holding the work.....	33
Speeds, feeds, and coolants.....	34
Milling operations.....	35
Indexing.....	36
Operating and safety precautions.....	37
V. Shaper.	
General.....	38
Parts and nomenclature.....	39
Cutting tools.....	40
Operating the shaper.....	41
Operating and safety precautions.....	42
VI. Grinders and grinding.	
General.....	43
Grinding wheels.....	44
Wheel selection.....	45
Grinding operations.....	46
Operating and safety precautions.....	47
VII. Power hacksaws.	
Description.....	48
	Page
APPENDIX I. Tables.....	163
II. Bibliography.....	183

SECTION I

GENERAL

	Paragraph
General.....	1
Laying out work.....	2
Precision measurements.....	3
Glossary.....	4

1. **General.**—*a. The machinist.*—(1) Machine shop work is generally understood to include all cold metal work by which an operator, using either power-driven or hand tools, removes a portion of the metal and shapes it to some specified form and size. Some cold metal work such as that done in sheet metal work and coppersmithing is not usually regarded as a machine shop operation.

(2) This manual explains the use of the power-driven machine tools with which every machinist must be familiar in order to per-

form efficiently the machine shop work commonly required for automotive repairs. A good machinist is thoroughly familiar with the tools covered in TM 10-590. He can read working drawings and knows how to lay out and set up jobs. In machining, as in all other motor repair shop work, all measuring tools must be correctly used if an operation is to be completed within specified tolerances. Taking proper care of all tools and equipment and using all the general safety precautions recommended here and those applying to each particular machine will improve the quality of the work turned out and will guard all workers against avoidable injury.

b. General safety precautions.—(1) Gears, pulleys, belts, couplings, ends of shafts having keyways, and other revolving or reciprocating parts should be guarded to a height of 6 feet above the floor. The guards should be removed only for repairing or adjusting the machine and replaced before operating it.

(2) Safety setscrews should be used in collars.

(3) It is extremely dangerous to start or attempt to operate a machine until familiar with how it works and its dangers have been fully explained by a qualified person. *Ignorance causes as many accidents in the machine shop as carelessness.* Knowledge and care will prevent most accidents.

(4) Loose or torn clothing, particularly loose or torn sleeves, a flowing necktie, or a flapping belt end is dangerous because it can easily catch in moving parts and draw in and crush the fingers, the hand or the arm, or other parts of the body. Running gears are among the most dangerous of mechanisms. All gearing should be well guarded. Never wear gloves when operating a machine except when absolutely necessary.

(5) Starting a machine while it is being adjusted or repaired is bad practice. Make sure that this does not happen.

(6) If metal chips, turnings, or shavings are removed with the hand, they may cause a serious cut. If the shavings are long, break them with the crooked end of a scratch awl or bent rod, and then brush them off the machine. Remove cast iron chips which break into small fragments with a brush.

c. Characteristics and types of machine tools.—The term “machine tool” commonly refers to any piece of power-driven equipment that drills, cuts, or grinds metals, or other materials. By using various attachments, some machine tools will perform two or all three of these operations; the machine tools actually hold and work the material; the operator merely guides their mechanical movements by the way in which he sets up the work and adjusts the gearing or linkage con-

trols which are integral with the machine. This manual explains the principles and operation of the following machine tools, which are ordinarily found in a completely equipped machine shop:

Drill presses.

Lathes.

Milling machines.

Shapers and planers.

Grinders.

Power hacksaws.

2. Laying out work.—*a. General.*—(1) “Laying out” is a shop term which means to mark guiding lines, circles, and centers on the curved or flat surfaces of the work. Such markings are, as a rule, transferred to the work from a blueprint or other drawing. The process is somewhat similar to making a mechanical drawing, but differs from it in the important respect that the position of every mark must be absolutely accurate if satisfactory results are expected. For that reason, a machinist keeps all his measuring and laying-out tools in good condition, sees that all his scribes, dividers, and punch points are sharp, and uses particular care to insure fine and accurate laying out.

(2) In scribing lines, draw the scribe across the work once, holding the point close to the straightedge; reruling thickens the line and leads to inaccurate work.

b. Reading working drawings.—Drawing is the language of the engineer, designer, and machinist; unless a mechanic can read working drawings, he cannot become a skilled machinist. There are certain standard methods of drawing views, lines, scales, sections, and other representations; their correct use and interpretation are not difficult to learn. A set of working drawings ordinarily consists of the following:

(1) *Assembly drawing.*—This shows the entire machine with all parts properly located. Such a drawing is usually made to a reduced scale, for example, one-quarter size or one-half size.

(2) *Detail drawings.*—These show each part of the machine separately and are often called “details.” A detail drawing should contain complete information for constructing the part, such as dimensions, material used, number of pieces, operations to be performed, tolerances, etc., and should consist of enough views to be read easily.

(3) *Sectional drawings.*—These show certain assembled portions of a unit, as if part of the stock had been sliced away. This is to illustrate its interior construction.

(4) *Bolt and screw lists.*—These lists tabulate all the necessary bolts, screws, etc.

(5) *Motion diagrams.*—These are provided when a machine has a number of complicated motions, to show the relations of certain centers to the motion of parts, velocity ratios, and directions of motion.

(6) *Views.*—Views are used by the draftsman to give the machinist three dimensions—length, breadth, and thickness—on a flat surface where only two can be shown. The usual method is to show a front view of the machine or part with other views grouped around it in the order of their names, as, for example, top view above, bottom view below, etc. Each view centers on a horizontal or vertical center line.

(7) *Lines.*—A full line on a drawing indicates a visible line or edge of the object. A dotted line indicates a hidden or invisible edge or line; a broken line (made up of alternate dots and dashes) indicates a center line; a dimension line is usually a full line with a break to provide room for figures and an arrowhead at each end to indicate the surface dimensioned. Section lines (cross hatching) are parallel lines ordinarily drawn at a 45° or 60° inclination, equally spaced; they are drawn across a surface to indicate that it is a section; various combinations and spacings of these section lines identify material as shown in table I. (For tables I to XIX, incl., see app. I.)

(8) *Scales.*—Whenever possible, drawings are made actual size, or full scale. When an object is too large to be conveniently represented full size, it can be reduced to half size, quarter size, eighth size, or sometimes smaller. *When working from drawings, always follow the dimensional figures; measurements should not be transferred from the drawing itself.*

(9) *Abbreviations.*—Table II shows the accepted standard abbreviations used on working drawings for machine shop use.

(10) *Sample drawings.*—Figure 1 is an example of an assembly or general drawing of a bench vise. Figure 2 is a typical detail drawing of the jaws of the same vise. TM 10-555 gives additional information on the subject of mechanical drawing and blueprint reading.

3. Precision measurements.—*a. General.*—Precision measurements are the basis of good machine-tool work. Before making any attempt to work with machine tools, review section III, TM 10-590, which deals with measuring tools, and be sure it is thoroughly understood. In addition to the tools mentioned therein, some other measuring instruments are in everyday use by machinists for laying out and

checking the accuracy of work. Remember that a job improperly or inaccurately laid out will not be worth machining.

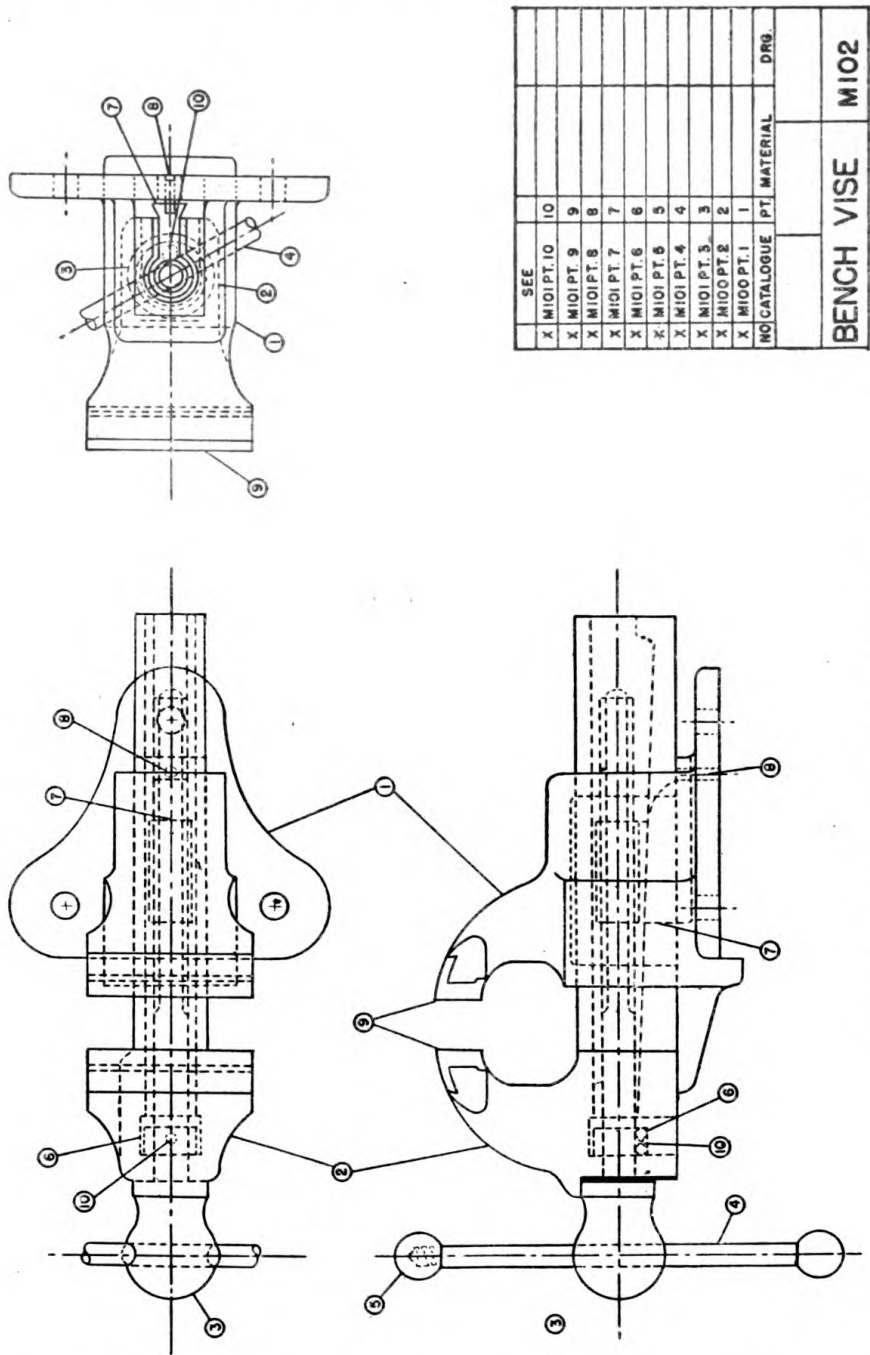


FIGURE 1.—Typical assembly drawing of bench vise for shop use.

b. Surface gages.—(1) The surface gage is used for laying out work for all classes of machine tools. The ordinary surface gage consists of a heavy base, an upright, and an adjustable scriber. In

the universal surface gage (fig. 3), the upright spindle is pivoted to the base so that it may be set at any angle; in some cases the base is grooved so the gage may be used on round work as well as flat surfaces. When the surface gage is used, both the work and the gage should be on a true plane surface such as a surface plate or machine table, and the scribe carefully adjusted to the desired height with a rule or combination set. Lines are drawn on the work by moving the gage along the plane surface while keeping the scribe in contact with the work. Most machinists prepare the work so that the scribed lines will show up distinctly; a rough or unfinished piece can be covered with chalk, or white lead mixed with turpentine; finished or bright surfaces should be copperplated by applying a thin coating of copper sulphate solution (1 ounce of copper sulphate in 4 ounces of water) with a brush or a piece of waste. In using copper sulphate solution the surface must be clean and free of all oil and grease, otherwise the solution will be dark in color and will not adhere to the surface. Shellac colored with purple dye is sometimes used for the same purpose.

(2) After scribing lines, it is good practice to place light prick punch marks along them at intervals so that their position may be relocated readily if the chalk or copperplating wears off. To obtain accurate lines, both the work and the surface gage must be on a true plane surface such as a surface plate. Any convenient surface such as a workbench or table may not be truly level and should not be used.

c. Keyseat rule.—A keyseat (or box) rule is the most convenient tool for drawing lines and laying off distances along curved surfaces. Figure 4 shows common types of keyseat rules being used to locate lay-out lines for a keyway on a cylindrical shaft. As an emergency measure, if a keyseat rule is not available, many machinists place round stock on the ways of the lathe and, using a pair of dividers with the ways as a guide, scribe a line on the work parallel to its axis.

d. Try square.—A try square consists of a beam and a blade fixed at right angles, the beam being much thicker and somewhat shorter than the blade. Such a square is used to draw lines at right angles to each other or to given surfaces; to erect and test perpendiculars to plane surfaces; to test the truth of a given surface at right angles to another surface; in short, wherever an accurate lay-out or test of 90° is required. When testing the relation of two surfaces, the beam should be pressed closely against the true surface and the blade brought carefully against the surface being tested.

e. Bevel.—A bevel is used to test the relation of lines and surfaces which are not at right angles to each other; actually, it is a try square

in which the blade is adjustable. Figure 5 shows its construction; its use is similar to that of the try square.

f. Protractors.—The bevel protractor (fig. 6) is a protractor provided with attached straightedges and is used in the machine shop to measure or scribe lines at any angle to each other. It has a disk grad-

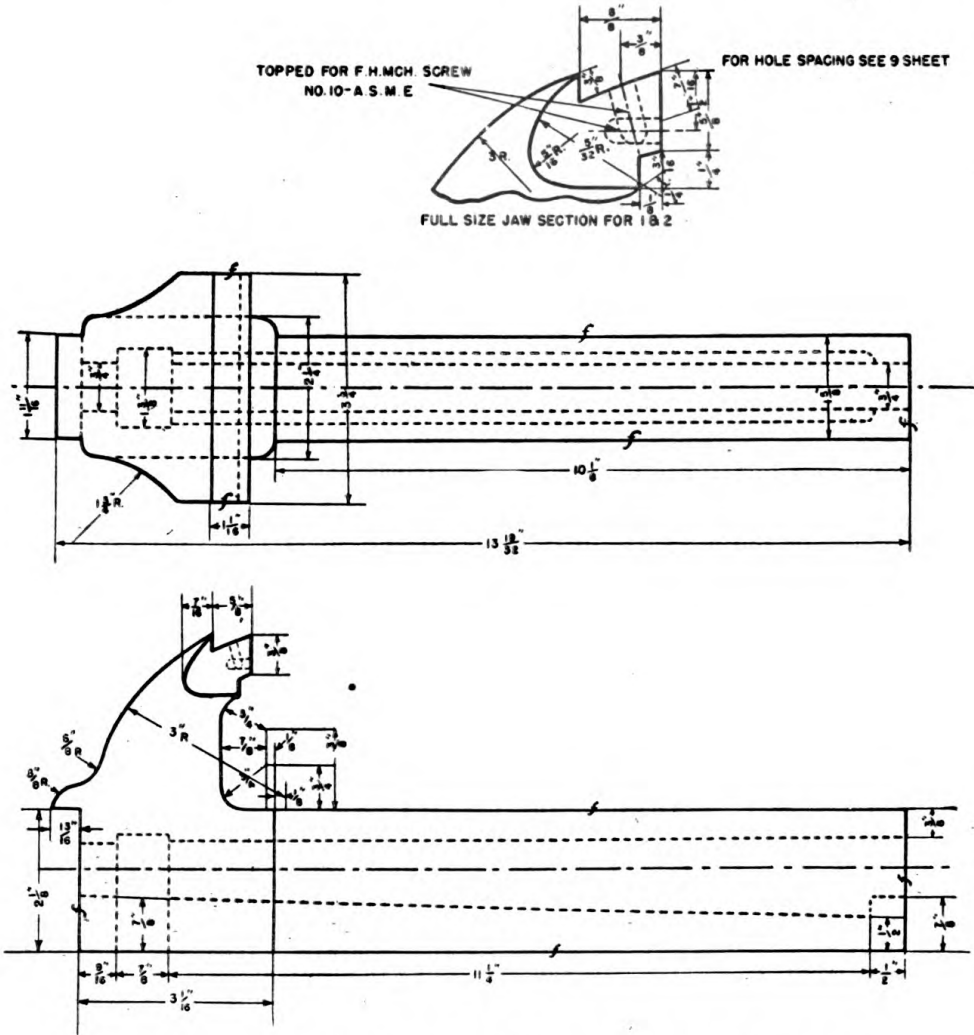
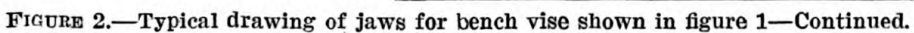


FIGURE 2.—Typical drawing of jaws for bench vise shown in figure 1.

uated in degrees through 180° (a half circle) and, if a vernier is provided, angles to one-twelfth of a degree (5 minutes) can be read.

g. Test indicators.—Test indicators are commonly used measuring tools which indicate small surface variations, usually in thousandths of an inch, upon a clearly graduated dial or arc. Various mechanisms based on a combination of short and long lever arms magnify any movement of the contact point of the indicator as it passes over a



tricity in a lathe, milling machine, or grinder; to indicate uniformity of height in a planer, shaper, boring machine, or milling machine; to indicate parallelism; as an attachment for the surface gage to test

or compare plane surfaces; and to test for alinement in any machine. Specific examples of their use are explained later in the manual.

h. Limits of tolerance.—Tolerance in machine tool work means an allowable variation from a dimension. For example, if a diameter is specified as 1 inch plus or minus 0.005 inch, it means that the diameter of the finished work may be 0.005 inch more or less than 1 inch.

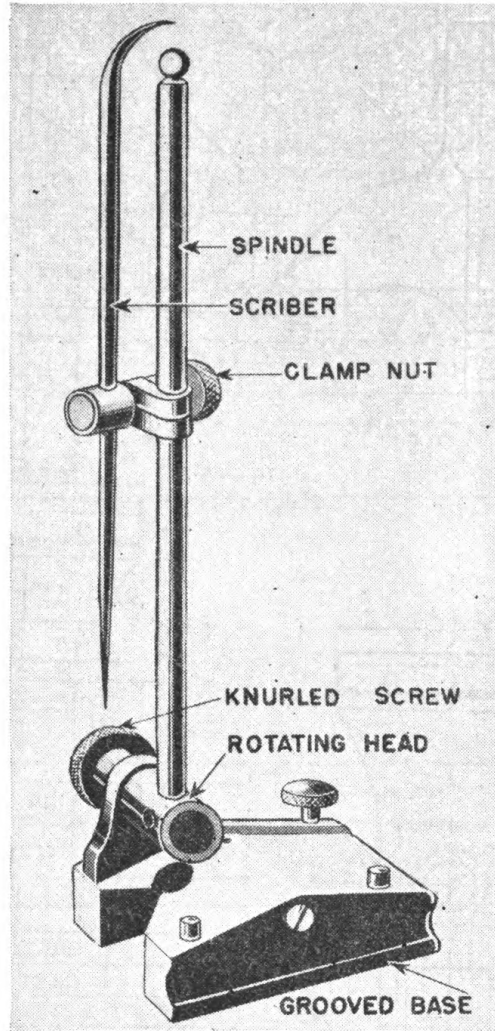


FIGURE 3.—Universal surface gage.

Limits of tolerance are best expressed as decimals; occasionally however they are written as fractions. A table similar to table III is available in most shops for converting the more commonly used fractions into decimals, and vice versa. In connection with the subject of tolerances, paragraph 27, TM 10-590, explains the use of fixed gages, especially those known as "go" and "no go" or limit

gages. Good automotive work demands working within close limits. Piston pins, pistons, cylinders, valves, steering knuckle pivots, crankshafts and camshafts are generally made to within about 0.001 inch tolerance. Antifriction bearings are often finished to within 0.0005 or a half-thousandth inch.

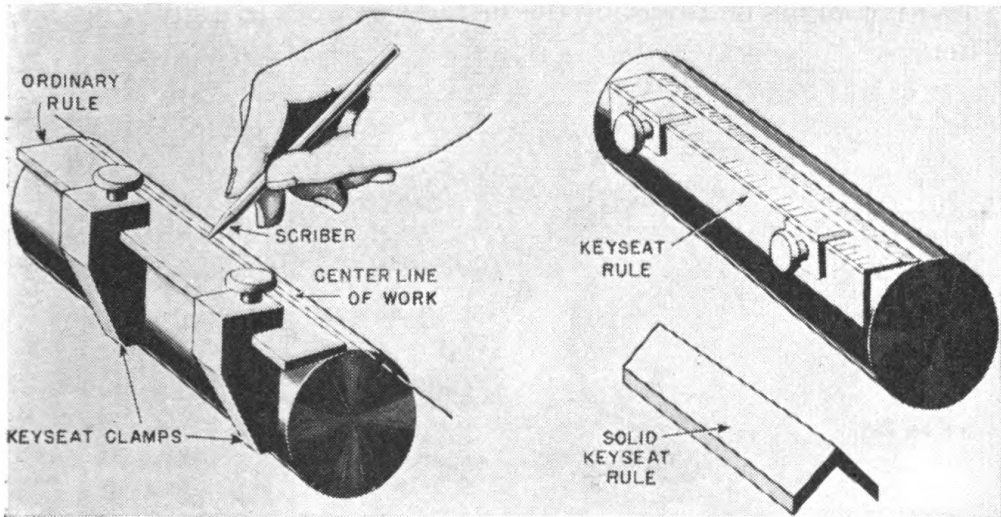


FIGURE 4.—Typical use of keyseat rules.

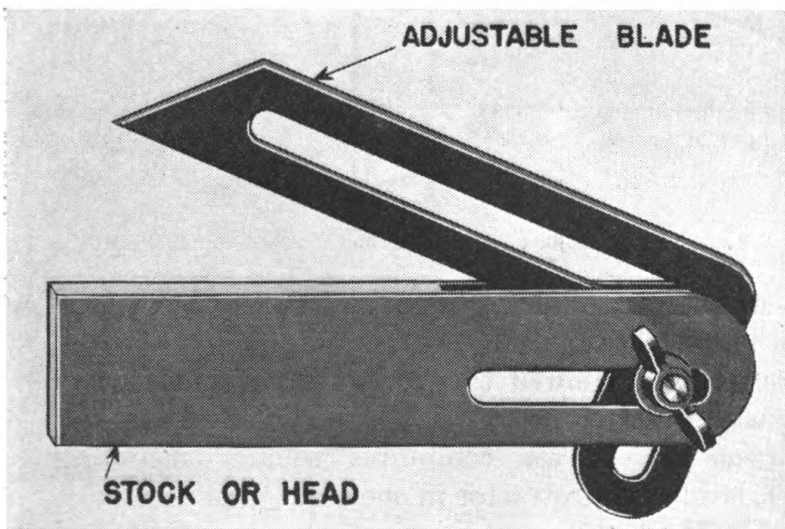


FIGURE 5.—Universal bevel.

4. Glossary.—For purposes of clarity and uniformity the following terms used in this manual are defined:

Arbor.—A shaft or bar for holding a cutting tool.

Arc.—A portion of a curved line; as the arc of a circle or ellipse.

Axis.—A center line. A line about which something rotates or is evenly arranged.

Bevel.—*a*. The angle which one surface or line makes with another when they are not at right angles; the slant or inclination of such surface or line.

b. An instrument consisting of a stock and adjustable blade for drawing angles or adjusting the surfaces of work to a given inclination.

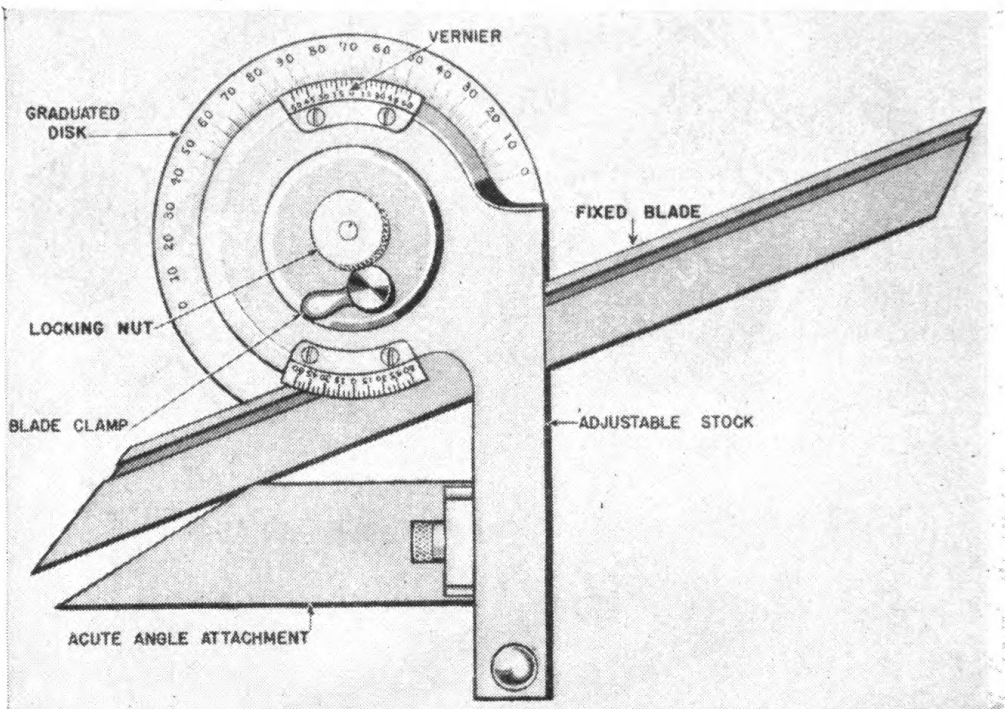


FIGURE 6.—Universal bevel protractor.

Chuck.—A device for holding work or a tool in a machine, especially in a lathe.

Clearance.—Space allowed to prevent interference between two or more parts of a machine.

Combination set.—A set, combining square (including centering square), level, and protractor in one tool.

Concentric.—Having the same center, as circles or spheres, one within another. Opposed to eccentric.

Countershaft.—An intermediate shaft which receives motion from a main shaft and transmits it to a working part.

Dead center.—A center which does not revolve.

Degree.—A 360th part of the circumference of a circle, or of a round angle. A degree is the principal unit of measure for arcs

THE MACHINIST

and angles. The degree is subdivided into 60 minutes and the minute into 60 seconds.

Eccentric.—Not having the same center; said of circles, cylinders, spheres, etc. Opposed to concentric.

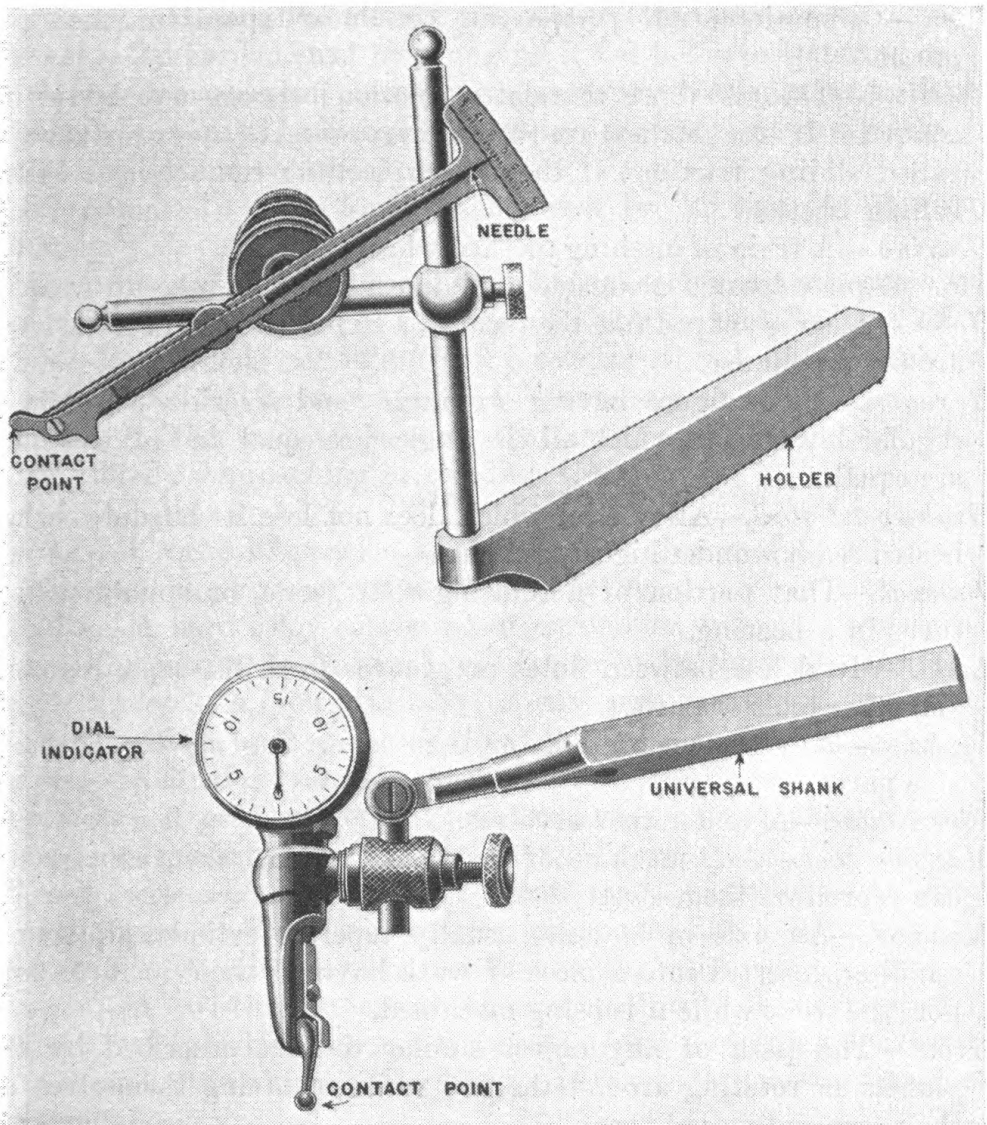


FIGURE 7.—Typical test indicators.

Feed.—The rate at which the cutting tool travels across the work or the work travels across the cutting tool. Measured in inches per revolution (lathe); inches per stroke (shaper); or feet per minute (milling machine).

Longitudinal (or long) feed.—Horizontal feed in the direction of the length of the machine.

Cross (or transverse) feed.—Horizontal feed in a direction across the width of the machine.

Vertical feed.—Feed in a vertical or up-and-down direction.

Fillet.—A concave or dish-shaped junction formed where two surfaces meet.

Fpm.—Circumferential, peripheral, or linear speed in feet per minute.

Friction.—The resistance to relative motion between two bodies in contact. If the bodies are in sliding contact, the resistance is called sliding friction; if they are in rolling contact it is called rolling friction.

Gearing.—A train of meshing toothed wheels.

Gib.—A piece located alongside a sliding member to take up wear.

Helix.—A curve shaped like the form of a string wrapped successively around a cylinder.

Hexagon.—Plane figure having six angles and six sides—usually a regular hexagon in which all the angles are equal and all the sides are equal.

High-speed steel.—Alloy steel which does not lose its hardness when heated red hot under high-speed cuts.

Journal.—That portion of a rotating shaft, axle, or spindle which turns in a bearing.

Land.—Metal left between flutes or grooves in drills, taps, reamers, and other tools or parts.

Linkage.—Any system of links or bars connecting moving or movable parts.

Live center.—A center that revolves.

Machine tool.—Any machine of that class which, taken as a group, can reproduce themselves.

Mandrel.—An axle or spindle, usually tapered, cylindrical, or expanding, inserted into a piece of work having a hole in it, to support the work while it is being machined.

Orbit.—The path of any object similar to that described by the planets in rotating around the sun while revolving themselves on their own individual axes.

Pawl.—A pivoted tongue or sliding bolt which engages notches in another part, such as a ratchet wheel, to permit motion in one direction and prevent it in reverse.

Periphery.—The circumference or perimeter of a circle or other closed, curved, or polygonal figure.

Pitch.—Distance apart of consecutive, equidistant things, as the distance from any point on a screw thread to a corresponding point on an adjacent thread, measured parallel to the axis; the distance from a point on a gear tooth to a corresponding point on an adjacent tooth, measured on the pitch line.

Planetary.—Having the motion of a planet (*see* Orbit).

Quench.—To harden steel by plunging it red hot into water or oil.

Ratchet.—A mechanism consisting of a gear with triangular teeth engaged by a pawl, which either moves the gear intermittently or locks it against backward movement.

Reciprocate.—To move forward and back (or up and down) alternately.

Relief or relieving.—The removing of metal, or the amount removed, to reduce friction back of the edge of a cutting tool; backing off.

Rpm.—Revolutions per minute; the number of revolutions, or complete turns, that an object makes in 1 minute.

Spiral.—*a.* The path of a point that moves around an axis while continually approaching or receding from it.

b. A helix.

Spline.—A series of parallel keys integral with a shaft, mating with corresponding grooves cut in a hub or fitting.

Stellite.—A hard alloy consisting of chromium, tungsten, and cobalt used for cutting tools, which will retain its hardness at red heat.

Surface plate.—A plate, usually of cast iron, having a true plane (flat) surface, used as a standard of flatness.

Taper.—Gradual decrease of thickness or width.

Template.—A gage, pattern, or mold, commonly a thin plate or board, used as a guide to form the work being shaped.

Throw (eccentric).—A crank; a crank arm; the radius of a crank or the crank radius of an eccentric, cam, or the like.

Tolerance.—Permissible variation from a dimension.

Torque.—A twisting or wrenching effort. Torque is the product of force multiplied by the radial distance from the center of rotation to the point at which it is applied. It is usually measured in foot-pounds.

Traverse.—A lateral or crosswise movement, as of the saddle of a lathe carriage, usually applied to the movement of a cutting tool or grinding wheel across the work, or to the movement of the work across a cutting tool or grinding wheel.

SECTION II

DRILL PRESS

	Paragraph
General	5
Types of vertical spindle drill presses	6
Drilling tools	7
Operation of drill press	8
Special drill press operations	9
Operating and safety precautions	10

5. General.—When holes are cut into or through metal by a rotating tool with cutting edges at its point, the operation is known as drilling, and the machine which holds, rotates, and feeds the cutting tool is called a drilling machine or more commonly a drill press. (For purposes of clarity the machine will be referred to throughout this manual as a drill press and the cutting tool as a drill.) While any machine with a rotating spindle can be used to drill holes, it is more usual to use a machine specially designed and equipped for the work. The average machinist will be concerned almost entirely with drill presses having vertical spindles, although horizontal spindle types are sometimes used for special production jobs. Most drill presses have the spindle bored to hold taper shank drills; straight shank drills can be used only in a chuck which in turn fits into the spindle.

6. Types of vertical spindle drill presses.—*a. Sensitive.*—(1) Figure 8 shows a typical drill press for use with drills up to and including $\frac{1}{2}$ inch in diameter on work which makes it necessary or advisable for the operator to “feel” what the cutting tool is doing. Smaller sensitive presses are available for mounting on the work bench or table. An electric motor drives the drill press illustrated through belts connecting the cone pulleys and the spindle. By moving the belt to different mating steps on the cone pulleys, three different spindle speeds can be obtained. (A drill press which obtains its power from a belt connected to a lineshaft has no motor.)

(2) When using the type of drill press illustrated, hold the work on either the square or round table or fasten it to the apron. The round table can be removed from its bracket and replaced with the crotch center or cup center. The crotch center is for locating cylindrical work so that it can be drilled through its diameter. The cup center approximately centers round or spherical work to be drilled axially. The size of a drill press is ordinarily given as the largest width or diameter of work which it will drill in the center; for example, a 14-inch drill press has half that distance, or 7 inches,

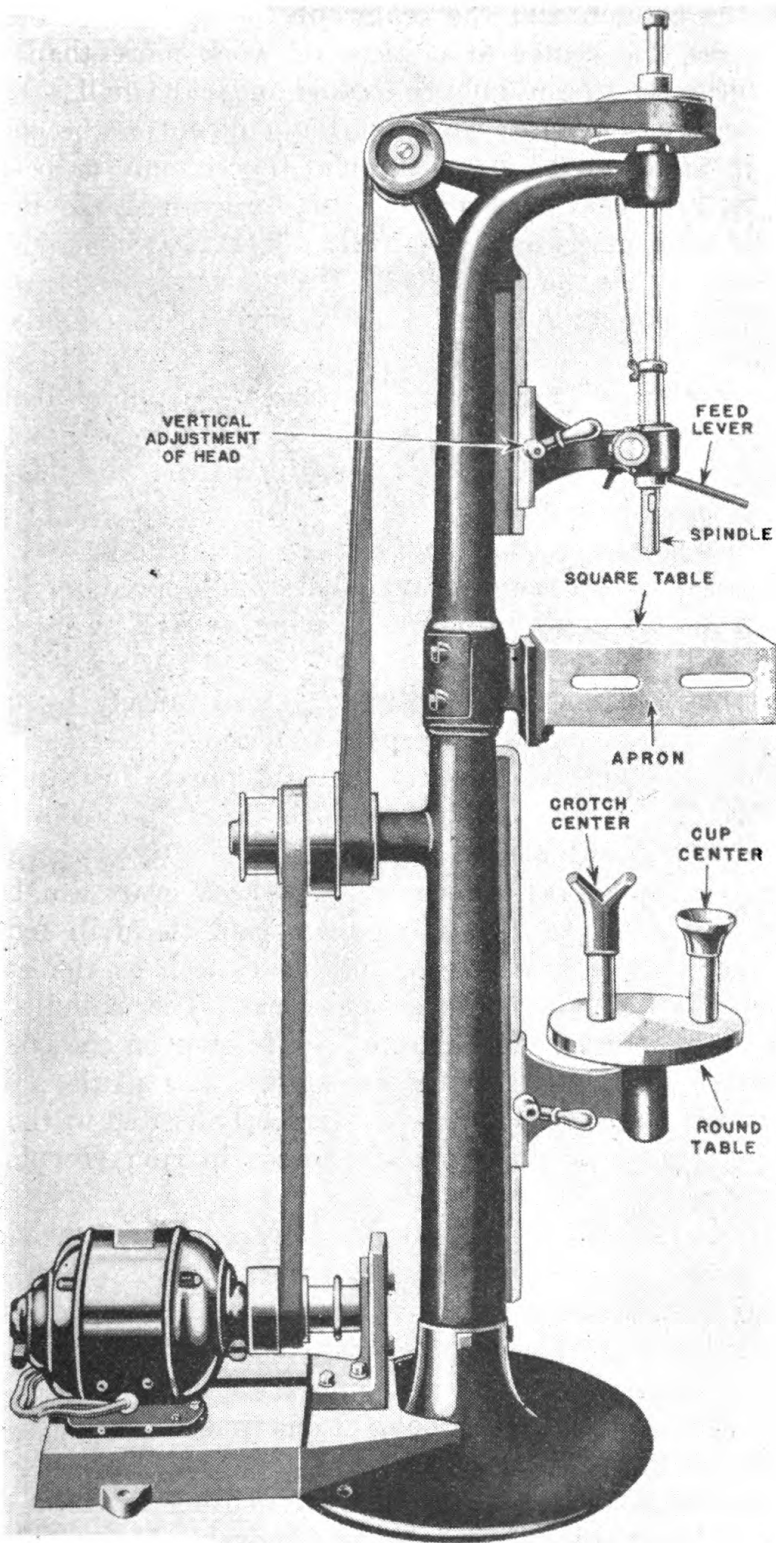


FIGURE 8.—Motor-driven sensitive drill press.

between the column and the center of the spindle. Therefore, in such a press, the center of a piece of work more than 14 inches in diameter or width could not be located under the drill.

(3) The spindle can be moved up and down on the column and the square table can be swung around the column, or be tipped at any angle, and held in place by bolts provided; the bracket for the round table moves only vertically. The peculiarity of sensitive drill presses is that they provide no mechanism to feed the drill into the work by power; the feed lever is used to control the feed by hand only.

b. Power feed.—(1) Figure 9 illustrates a popular 21-inch, back-gear, power-feed drill press, capable of handling drills $1\frac{1}{2}$ inches in diameter and smaller. This press differs from the sensitive type in that—

- (a) It has a heavier frame and moving parts and greater capacity;
- (b) It has a greater range of available spindle speeds;
- (c) Its spindle feeds by power gearing as well as by handwheel or lever; and
- (d) It has greater spindle torque, gained largely by use of the back gears, with accompanying speed reduction.

(2) Both the table and base of the drill press illustrated are provided with T-slots so that heavy work can be bolted or clamped firmly to them; the table can be moved vertically on the column by means of the table traverse screw. The back gears can be quickly engaged or disengaged by sliding them, and the drill fed into the work either by the power feed, the handwheel, or the hand lever, which operates on the ratchet principle. The spindle speed is changed by moving the belts from step to step on the cone pulleys and engaging or disengaging the back gears. As a rule, a drill press of any type has an etched plate conveniently visible to the operator which specifies its available spindle speeds in rpm (revolutions per minute).

c. Multiple spindles.—A drill press is sometimes provided with more than one spindle. In smaller presses of the vertical type, the spindles are fixed in their relative positions and are not intended to be operated simultaneously, the work passing from one spindle to another. This is known as a “gang driller.” The true multiple-spindle machine drills several holes at one time in any position within the machine’s limits of adjustment.

d. Radial drillers.—Another form of drilling machine called the radial type has a spindle mounted in a movable horizontal arm, and a tilting table. The spindle can be moved along the arm, and the

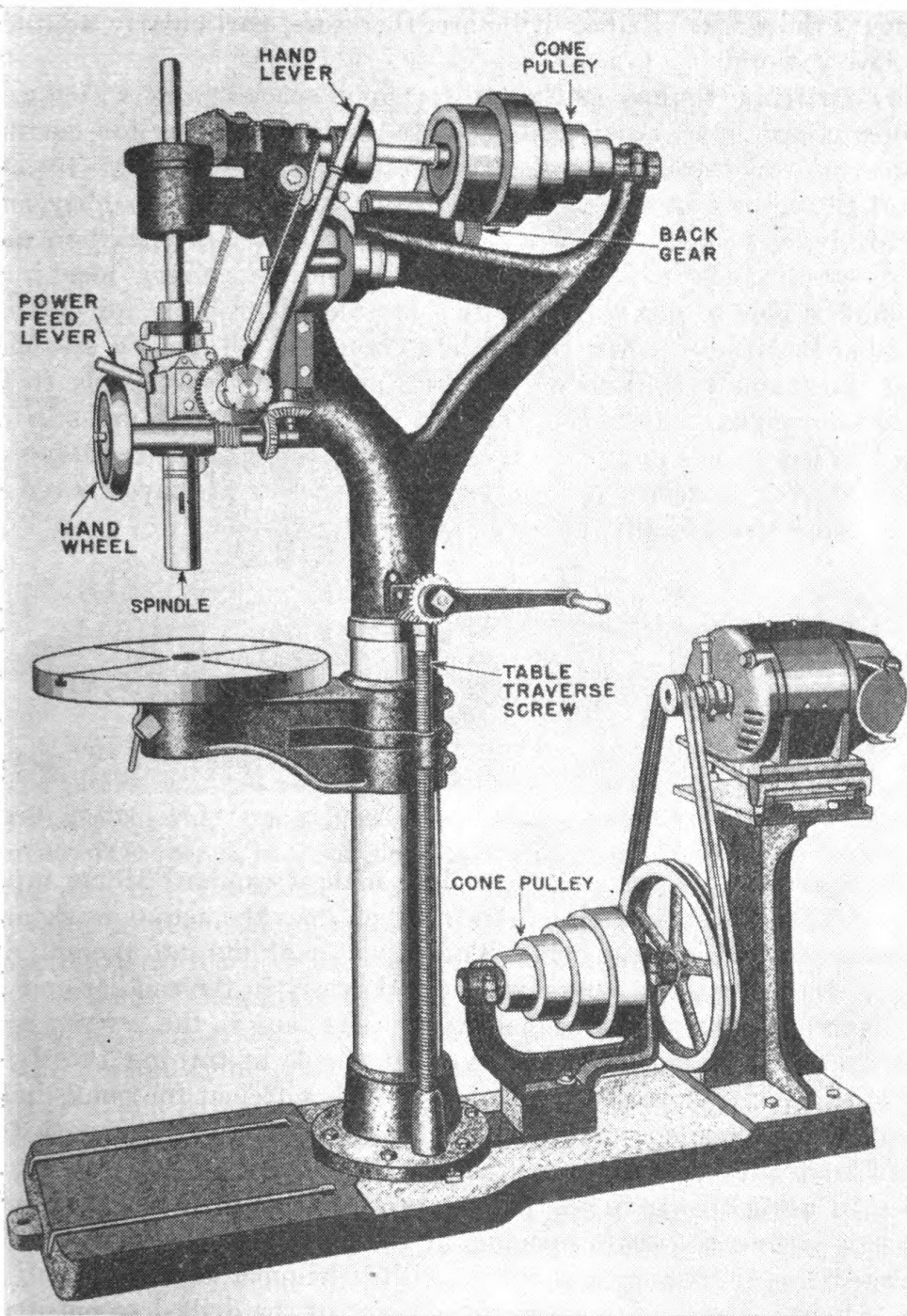


FIGURE 9.—Motor-driven power-fed drill press.

arm moved vertically on and rotated around a stationary column. Holes can be drilled in any location within reach of the arm without moving the work. Radial drills are, therefore, particularly adapted to heavy work.

7. Drilling tools.—*a.* Except for some special work which will be explained later, twist drills are used in drill presses for cutting holes into or through metal. Paragraphs 11 and 12, TM 10-590, treat the subject of twist drills in detail; review it thoroughly and carefully, especially the portions covering nomenclature, carbon and high-speed steels, cutting angles, and grinding. Every machinist should be able to grind (sharpen) a twist drill properly for cutting the particular metal with which he is working. Table IV specifies and illustrates recommended grinding angles for twist drills to be used on common materials. The only difference between twist drills used in hand tools and those used in drill presses is in the shape of their shanks; standard drill press spindle sockets are taper bored to hold taper shank drills.

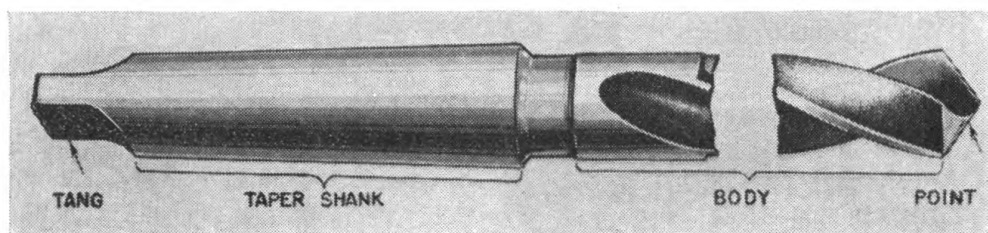


FIGURE 10.—Taper shank twist drill.

b. Figure 10 illustrates a twist drill with a standard Morse taper shank. It consists of a fluted body which does the actual work and the shank by which it is held. This taper shank fits into the spindle of the drill press; the tang slips into a keyway in the smaller end of the spindle. The positive engagement of the tang in the keyway aids the frictional engagement of the taper shank in turning the drill. The frictional resistance of the taper alone is sufficient for small drills but not for large ones. The tang also makes it easy to remove the drill from the socket by means of a drill drift (fig. 11).

c. In using the drift, the correct procedure is to insert the small end of it into the drift opening in the spindle with the flat edge against the drill tang, and tap it with a hammer until the drill is forced out. Use care to see that the point of the drill does not drop on the work of the press table.

d. Be sure that both the inside of the spindle socket and the shank and tang of the drill are smooth, clean, and free from nicks or burs

before fitting the drill into the socket. If they are not, a poor fit will result, which may break the tang by putting too much of the turning strain on it.

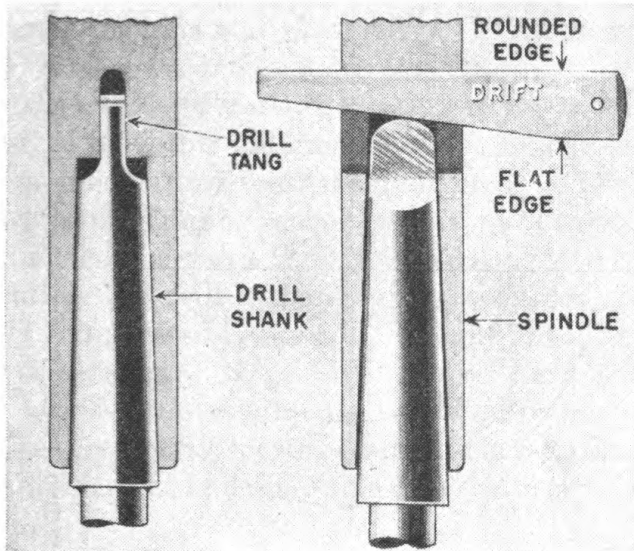


FIGURE 11.—Drill drift.

8. Operation of drill press.—a. Laying out and centering work.—

(1) Laying out work for drilling operations consists of locating the centers of the holes to be drilled. Unless the holes are to match other holes or fixed studs, approximate lay-outs with a chalk pencil and a steel rule are generally accurate enough. When special accuracy is required, however, lay out the location of each hole as shown in figure 12. The usual procedure is first to prepare the work with a light application of copper sulfate solution, and scribe two or more lines on it which intersect at the point which is to be the center of the hole. Then mark this intersection lightly with a prick punch. See that the mark is *exactly* at the point where the lines intersect, using a magnifying glass if necessary.

(2) Use a pair of dividers to scribe a circle on the work having a diameter equal to that of the hole, as shown in the center sketch (fig. 12). Make small indentations, known as "witness marks," on the circumference of the circle with a prick punch. If a check is required to see that the lay-out has been followed after the hole has been drilled, a second circle should be scribed outside the first one, as shown at the right in figure 12. Then enlarge the center prick punch mark with a center punch; if this is done before the dividers are used, the dividers will have a tendency to "walk."

(3) To center the drill properly in the work, place the drill point in the center punch impression and start drilling until the point has made an impression somewhat larger than the center punch mark;

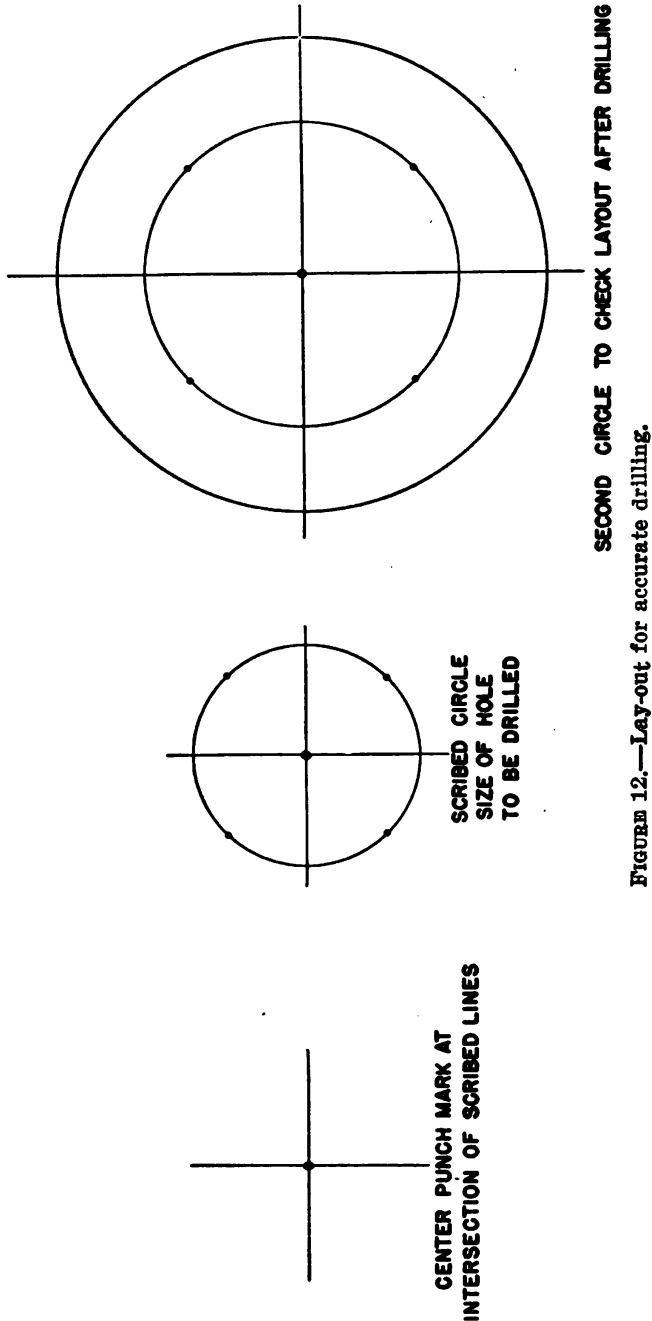


FIGURE 12.—Lay-out for accurate drilling.

then lift the drill from the work and see if this impression is concentric with the scribed circles. For various reasons, the drill as a rule will not be correctly centered the first try. To draw it back to

the center desired, cut a nick in the impression with a round-nosed chisel on the side toward which the drill is to be drawn, as shown in figure 13. This operation must be performed before the drill point has enlarged the hole to full diameter. If the drill has been properly centered, the hole will efface all of the smaller scribed circle, leaving half of each prick-punch mark showing.

b. Holding the work.—Before attempting to use a drill press, some provision must be made for holding the work solidly in place on the table of the machine. Most sensitive drill presses have tables which provide no means of clamping or bolting the work to them. Machinists generally fasten the work tightly in a heavy drill vise (fig. 14) and center it under the drill by hand; since work done in such a drill press is usually comparatively light, the weight of the vise is sufficient to hold the work steady. Larger drill presses have slotted tables so that work of considerable weight can be bolted or clamped firmly to them, and if an accurately located hole is desired, the work should always be firmly fastened to the table by whatever means its shape and size permit.

c. Speed.—(1) Correct speeds and feeds are essential to satisfactory drill press operation. The speed of a drill is usually given as the speed of its circumference, or peripheral speed in feet per minute (fpm). Actually, peripheral speed is the distance a drill would roll in one minute along a flat surface; for example, if a drill were rolled 70 feet along the floor in 1 minute, its speed would be 70 fpm.

(2) The speed at which a drill should be rotated in a drill press depends on the diameter of the drill and the material being drilled; no definite rules can be set up for regulating it, although various tables are available which give recommendations based on practical experience. Table V gives an approximate guide to satisfactory speeds under normal machine and material conditions. However, experience and sound judgment are necessary to determine the best speed for each operation; often the revolutions per minute of the drill press spindle necessary to obtain the desired peripheral speed of the drill being used must be known.

(3) If a table such as table V is available, it can be referred to; if such a table is not at hand, however, the following method of changing fpm and rpm is easy and quick. If the peripheral speed per minute of the drill is divided by its circumference, the result will be rpm. Note that if the peripheral speed of the drill is measured in feet per minute its circumference must be measured in feet. It is more convenient to change the peripheral speed of the drill to inches per minute and divide by its circumference in inches. For example, sup-

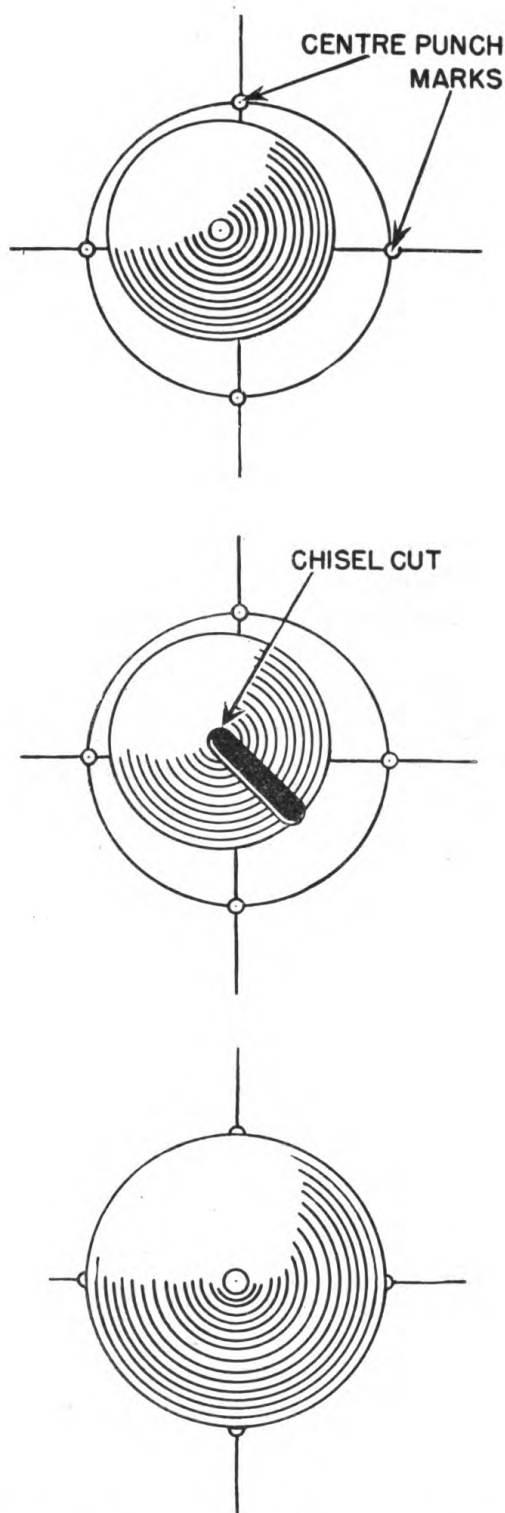


FIGURE 13.—Drawing back drill to correct center.

pose a machinist wants to drill in cast iron at 70 fpm with a $\frac{1}{4}$ -inch drill; 70 feet per minute is 70 times 12 or 840 inches per minute; 3.1416 (π) times the diameter of the drill ($\frac{1}{4}$ inch) equals its circumference, 0.7854 inch; 840 divided by 0.7854 equals 1069.51. The spindle of the drill press should therefore rotate at as nearly 1,070 rpm as possible. If the exact number of rpm desired is not obtainable on the press, it is generally better to use too much speed than not enough.

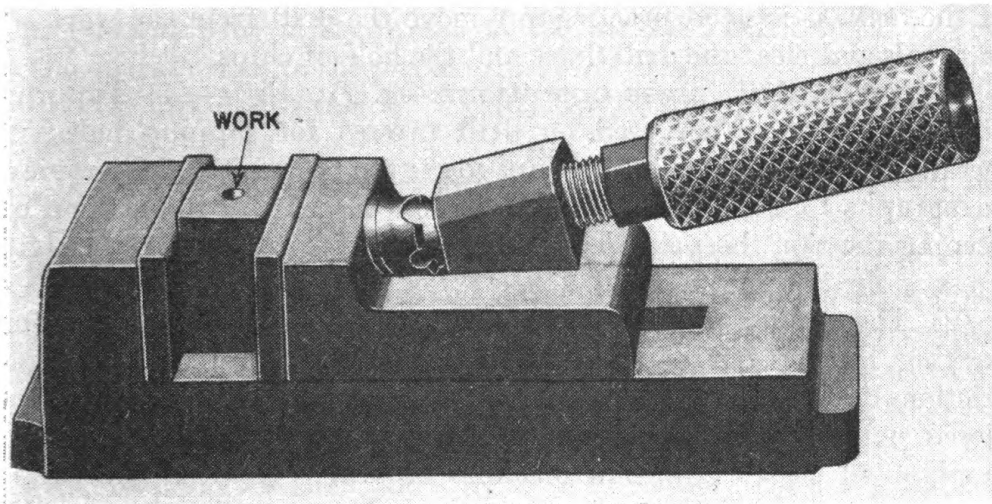


FIGURE 14.—Drill vise.

d. Feed.—Feed is the distance expressed in fractions of an inch a drill travels into the work during each revolution. As with speed, the best feed depends on the size of the drill and the material being drilled. The general rule is to use a feed of 0.001 to 0.003 inch per revolution with drills smaller than $\frac{1}{8}$ inch; 0.004 to 0.005 inch with drills from $\frac{1}{8}$ to $\frac{1}{4}$ inch; 0.005 to 0.008 inch with drills $\frac{1}{4}$ to $\frac{1}{2}$ inch; 0.008 to 0.013 inch with drills from $\frac{1}{2}$ to 1 inch; and 0.011 to 0.016 inch with drills larger than 1 inch. Alloy and hard steels should generally be drilled at a lighter feed than given above; cast iron, brass, and aluminum can usually be drilled with a somewhat heavier feed. Tables similar to table V are often available in the shop; if so, refer to them and be guided accordingly. In any drilling operation, use a very light feed when starting the hole; if the drill press has a power feed, the drill should be started into the work by hand before engaging the power feed.

e. Lubricants.—To maintain the speeds and feeds recommended for various drilling operations, use a suitable cutting compound or lubricant to obtain maximum cutting efficiency and avoid overheating either

the drill or the work. Experience has shown the following lubricants to be satisfactory for drilling:

Hard and refractory steel—turpentine, kerosene, or soluble oil.

Soft steel and wrought iron—lard oil or soluble oil.

Malleable iron—soda water.

Brass and cast iron—dry (no lubricant).

These recommendations apply either to carbon steel or high speed steel drills. Apply freely whatever lubricant is being used and, if the hole being drilled is much deeper than four times the diameter of the drill, it is good practice to remove the drill from the work at intervals and clear the drill flutes and the hole of chips.

9. Special drill press operations.—*a. Tapping.*—(1) Tapping attachments are often used on drill presses for tapping holes in metal. Before making any attempt to use the tapper, carefully review paragraphs 15, 16, 17, and 18, TM 10-590. Except for the method of turning the tap, the principles are the same for tapping in the drill press as for tapping by hand.

(2) The tapper must be equipped with a reversing device for backing the tap out of the hole, and usually the tap is held in a friction head that will slip when the tap strikes the bottom of a closed hole. Table VIII gives the recommended speeds for commonly used sizes of taps in machine tapping. No feed control is necessary; taps pull themselves into the hole at the correct rate, the same as a screw.

b. Reaming.—Reamers can be held, turned, and fed by a drill press when special accuracy of size and roundness of a hole are desirable. The principles are the same as set forth in paragraph 14, TM 10-590. Table VIII gives recommended speeds and feeds for commonly used sizes of reamers for machine operations.

c. Counterboring and countersinking.—Paragraph 13, TM 10-590, discusses the principles of, and tools used for, counterboring and countersinking holes. Countersinks and counterbores are frequently used in a drill press. A common job is the use of a 60° countersink, or a combination center drill and countersink, for drilling center holes in work to be set up between centers in a lathe.

d. Lapping (par. 46f).—This is the process of finishing the surface of a piece of work by grinding it with a softer metal with some kind of an abrasive between them or imbedded in its surface. The softer piece of metal is called a “lap.” A lapping operation sometimes done in a drill press is finishing the inside of a hole within a very close tolerance as to size and roundness.

e. Spot facing.—Spot facing is the process of machining a circular surface around the top of a hole to form a true bearing for a washer or collar. A counterbore is preferably used for this operation.

10. Operating and safety precautions.—*a.* In shop drill press operations sooner or later some difficulties will arise; few of them, however, are new, and all of them have some remedy. Table VI lists the most common drilling troubles, their causes, and the ways to correct them.

b. The following safety precautions if observed will prevent injury to the operator and to the machine:

(1) Oil the drill press as directed by the manufacturer's manual at least twice every day.

(2) When drilling a hole, do not let the spindle feed beyond its limit of travel.

(3) A short piece of work held with the hand while drilling is likely to be jerked away suddenly and cause a painful and often serious accident by cutting or bruising the hand. Work should be fastened firmly in a vise clamped to the table or screwed to a suitable fixture.

(4) Hammering on any part of a drill press or dropping tools on its table or base will damage it.

(5) Keep fingers, head, and arms away from tools that are in motion.

(6) Waste or rags used to clean tools that are in motion may be caught in the moving parts and drag the fingers or hand after them.

(7) Loose-fitting clothing may catch in a revolving drill. Most shops, safety engineers, and insurance companies require workmen to have short sleeves and close-fitting overalls.

SECTION III

SCREW-CUTTING ENGINE LATHE

	Paragraph
General	11
Parts and nomenclature of lathe	12
Cutting tools	13
Grinding lathe tools	14
Leveling and setting up lathe	15
Plain turning (work between centers)	16
Chucked work	17
Turning and boring tapers	18
Drilling, tapping, and reaming	19
Screw cutting	20
Eccentric turning	21
Special set-ups	22

	Paragraph
Boring.....	23
Special lathe work.....	24
Safety precautions.....	25

11. General.—*a. Principles of lathe.*—The lathe is a machine for removing metal from revolving work by means of suitably formed cutting tools. The cutting tools are made of hardened and tempered steel, or are of steel tipped with such special alloys as tungsten carbide or stellite. The usual lathe has back gears to provide the low spindle speeds and high torque required for heavy cuts on large-diameter work; change gears and a lead screw for cutting threads; and power longitudinal and cross feeds. When so equipped it is known as an engine lathe. This is the type used in Army motor maintenance and repair shops. In the hands of a competent operator, a lathe is the most versatile of all machine tools. It is so important that every machinist must thoroughly understand its operation.

b. Types, sizes, and drives of lathes.—(1) The general purpose screw-cutting engine lathe is the most common type of lathe in the average machine shop, and its operation will be explained in detail. Other types often used in production work are—

(a) The vertical turret lathe which is the fastest machine for short or heavy work.

(b) The horizontal turret lathe or screw machine used extensively in the high speed production of duplicate parts. Most such lathes are equipped with a pump and metal basin for the automatic application of a coolant to the work.

(c) The production lathe and the automatic screw machine used for the rapid production of small duplicate parts.

(d) Special purpose lathes made to perform one specific operation, such as a brake drum lathe or a crankshaft lathe. In these machines, certain features of general purpose lathes are omitted and certain others added to make them as nearly ideal as possible for a special operation.

(2) The size of a general purpose lathe is ordinarily given as the swing over the bed and the length of the bed. For example, a 16-inch by 8-foot lathe (fig. 15) is one having a swing over the bed sufficient to take work up to 16 inches in diameter and having a bed length of 8 feet. The swing over the saddle is less than the swing over the bed, and the maximum distance between centers is less than the length of the bed. These figures must be considered carefully, as they limit the size of work that can be machined between centers. As a rule, a

lathe should have a swing and bed length at least 10 percent greater than the largest piece of work which will be machined in it.

(3) Two types of drives for lathes are in common use: the direct or individual motor drive; and the group drive. Army lathes are practically all individual motor driven, which means that each lathe has its own electric motor, mounted either underneath the headstock or on a separate pedestal back of the machine. Lineshaft drive lathes are driven by belts connected through cone pulleys and counter-shafts to a lineshaft, which in turn is rotated by some prime mover. One lineshaft can drive any number of lathes within its capacity. Each type of drive has advantages and disadvantages but neither affects the fundamental operating principles of the machine.

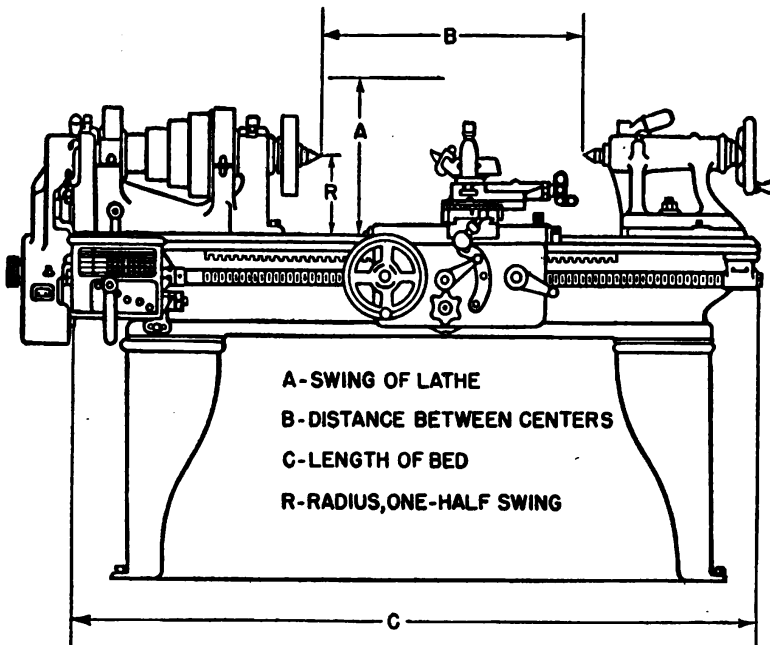


FIGURE 15.—Size and capacity of a lathe.

c. Range of operations of the lathe.—The lathe in the hands of an experienced operator is the most versatile of all machine tools. It will machine work round, concentric, eccentric, or tapered; cut inside or outside plain or tapered screw threads; drill, bore, ream, and tap straight or tapered holes; it is convenient for filing, polishing, or lapping, and for winding springs or electrical coils. By using attachments and adapters it can perform milling and grinding operations. It will do all work within close tolerances. It is an extremely useful machine in any automotive repair shop.

12. Parts and nomenclature of lathe.—*a. General.*—Figure 16 shows a popular type of back-geared motor-driven screw-cutting bench

lathe. The only essential difference between this lathe and the one shown in figure 15 is that the latter has floor legs. All its parts and controls are clearly indicated and named; this illustration should be studied along with this paragraph until the function of every part shown is understood. Lathes of the same type furnished by various manufacturers differ only in refinements of detail, not in principles of operation. Lathes of the same type usually have similar parts and

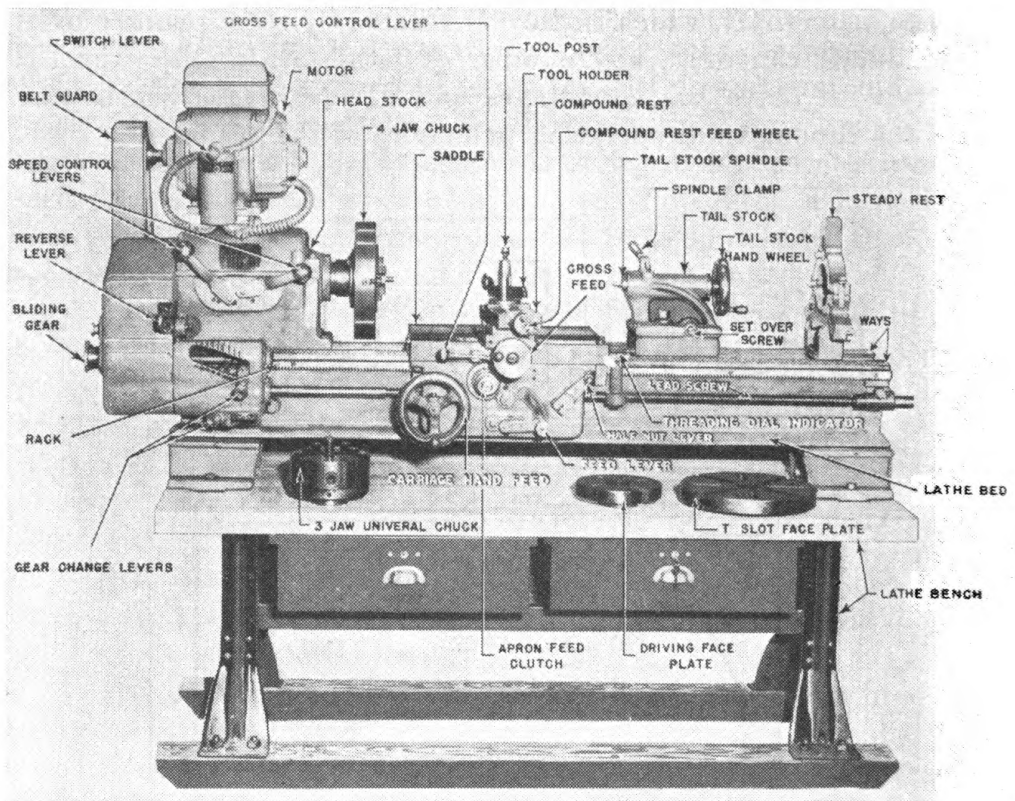


FIGURE 16.—Back-geared screw-cutting bench lathe.

controls. It is essential, however, that the manuals issued by the various lathe manufacturers for their own machines be studied before any except the simplest work is undertaken.

b. Bed.—The lathe bed is the foundation on which the lathe is built. It is a single iron or semisteel casting consisting of two heavy sides rigidly supported by cross members. The flat ways and V-ways which guide the carriage and align the headstock and tailstock are machined on the bed and hand-scraped to a high degree of accuracy. Figure 17 illustrates a typical lathe bed with the flat way and V-ways clearly shown. The ways should be kept clean at all times, and frequently wiped with an oil soaked rag. *Using the bed as an anvil for*

driving arbors in and out, or as a bench for hammers, wrenches, chucks, or any tools will ruin it.

c. Cone headstock.—(1) The complete headstock of the lathe consists of the headstock casting, which is located rigidly on the ways at the operator's left and contains the spindle, cone pulley, back gears, and controls. Figure 18 shows a back-geared headstock with the gear guards removed. (Lathes are available which do not have back gears, but they are not commonly used for general purposes.) Power is transmitted to the spindle from the motor or countershaft through a belt over the cone pulley to the back gears and then to the spindle, or

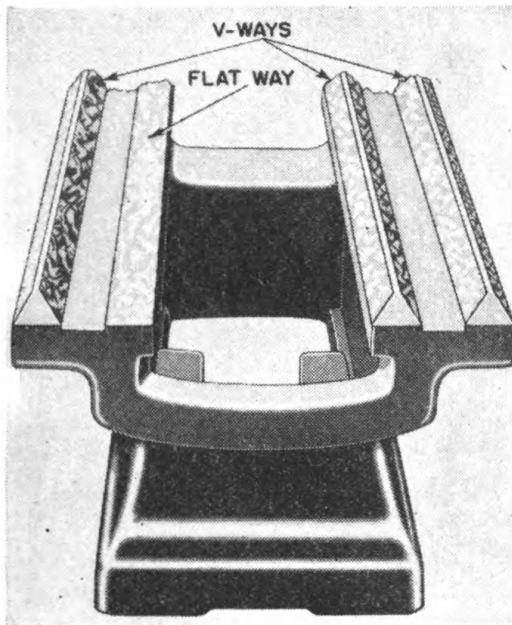


FIGURE 17.—Lathe bed.

directly to the spindle if the back gears are not in mesh, so that a choice of six or eight spindle speeds is usually available in either direction of rotation. The spindle is hollow, threaded on one end to hold a faceplate, chuck, or other attachment, and taper bored to hold the live center rigidly.

(2) The back gears are engaged or disengaged by the back gear lever; the gears are engaged when this lever is toward the front of the lathe. To disengage the back gears, put the back gear lever toward the back of the machine and have the bull gear lock in the "in" position. This locks the bull gear and the spindle directly to the cone pulley. To drive the spindle through the back gears, first see that the bull gear lock is in the "out" position. If the back gears are

disengaged and the bull gear lock in the "out" position, no power will be transmitted from the motor to the spindle.

(3) Spindle speeds are changed by moving the belt from the countershaft from one step of the cone pulleys to another, and by engaging or disengaging the back gears. Most lathes have a plate conveniently visible to the operator which tabulates the position of the headstock controls to secure the available spindle speeds in rpm. If the lathe does not have such a plate, consult the manufacturer's manual for directions.

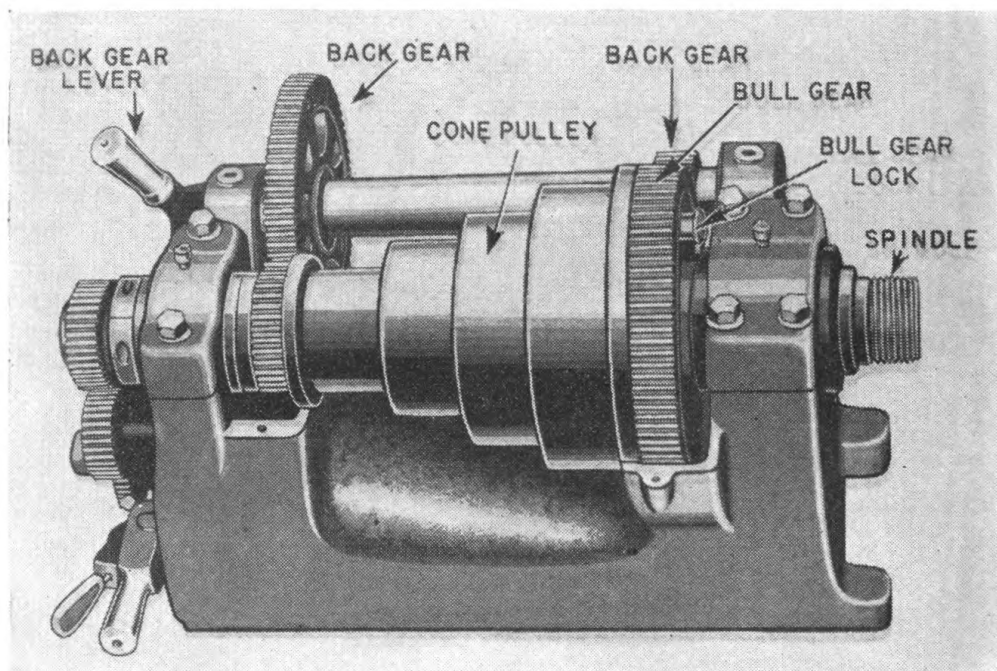


FIGURE 18.—Back-geared headstock.

d. Geared headstock.—Some lathes are equipped with headstocks which have no cone pulley. In these, the spindle speeds are changed by means of gears which are selected by moving levers on the front of the headstock. A chart specifies the positions of the levers to obtain the spindle speeds available.

e. Tailstock.—The tailstock of the lathe is a casting which moves along the ways and holds the dead center rigidly in a taper. It can be locked at any location on the ways by tightening the clamp bolt nut. By turning the handwheel the operator can move the spindle, which holds the dead center, lengthwise in the casting, and by pulling the binding lever or spindle clamp forward, can lock the spindle in any position. A screw adjustment is provided to set over the tail-

stock top for taper turning. Figure 19 shows a typical lathe tailstock with its parts and controls clearly indicated.

f. Carriage.—The lathe carriage, consisting of the saddle and apron, is the movable part which slides along the ways between the headstock and tailstock. The saddle is bridged across the bed in the form of the letter H, and carries the compound rest and tool post. Figure 20 illustrates a typical lathe carriage with the apron, saddle, compound rest, and tool post in their usual relative positions.

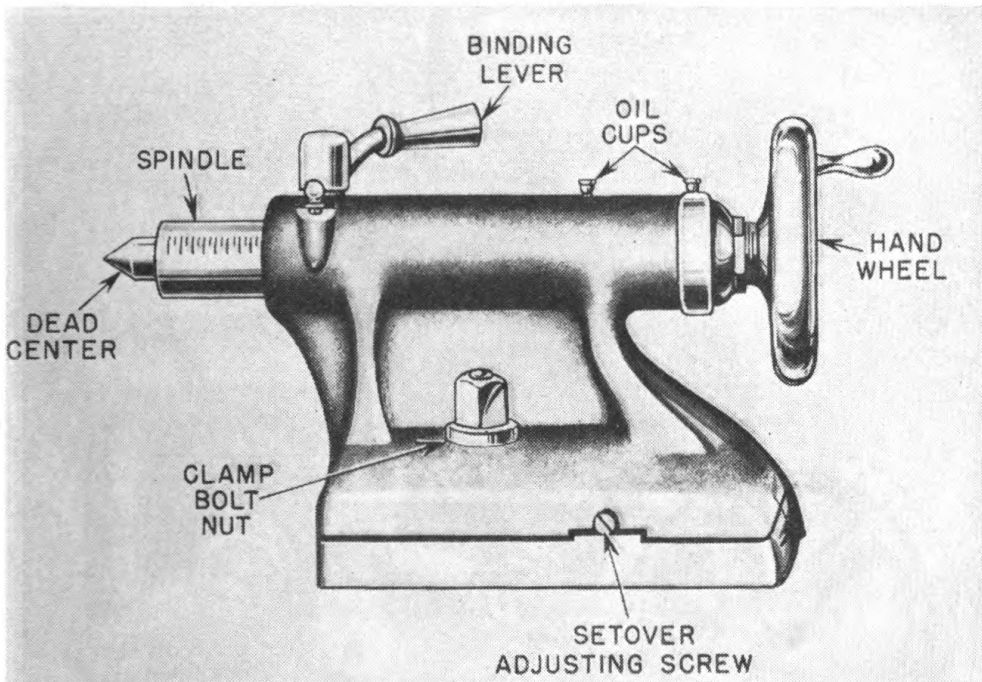


FIGURE 19.—Tailstock.

g. Saddle and compound rest.—Figure 21 shows a saddle and compound rest of the type ordinarily used on modern lathes. The purpose of the compound rest is to provide the cutting tool with a solid support which is readily and exactly adjustable to any position desired. The cross feed screw handle feeds the rest base back and forth across the bed on the saddle; the compound rest feed screw handle moves the compound rest back and forth on the base at whatever angle the swivel is set. Both these screws have collars graduated in thousandths of an inch so the feed can be accurately controlled. The compound rest swivel is graduated in degrees. It can be set at any angle by the swivel set screws for turning or boring tapers or bevels, and for some screw cutting operations. The cutting tool is held in the tool post, usually by means of a tool holder as shown in figure 21.

h. Apron.—(1) The apron of the lathe consists of a casting which contains the mechanism and controls for moving the carriage and engaging and disengaging the automatic power longitudinal and cross feeds, and the lead screw for thread cutting. Figure 22 shows the front of a typical lathe apron; figure 23 shows the back of the assembly and its principles of operation. It is very important that the

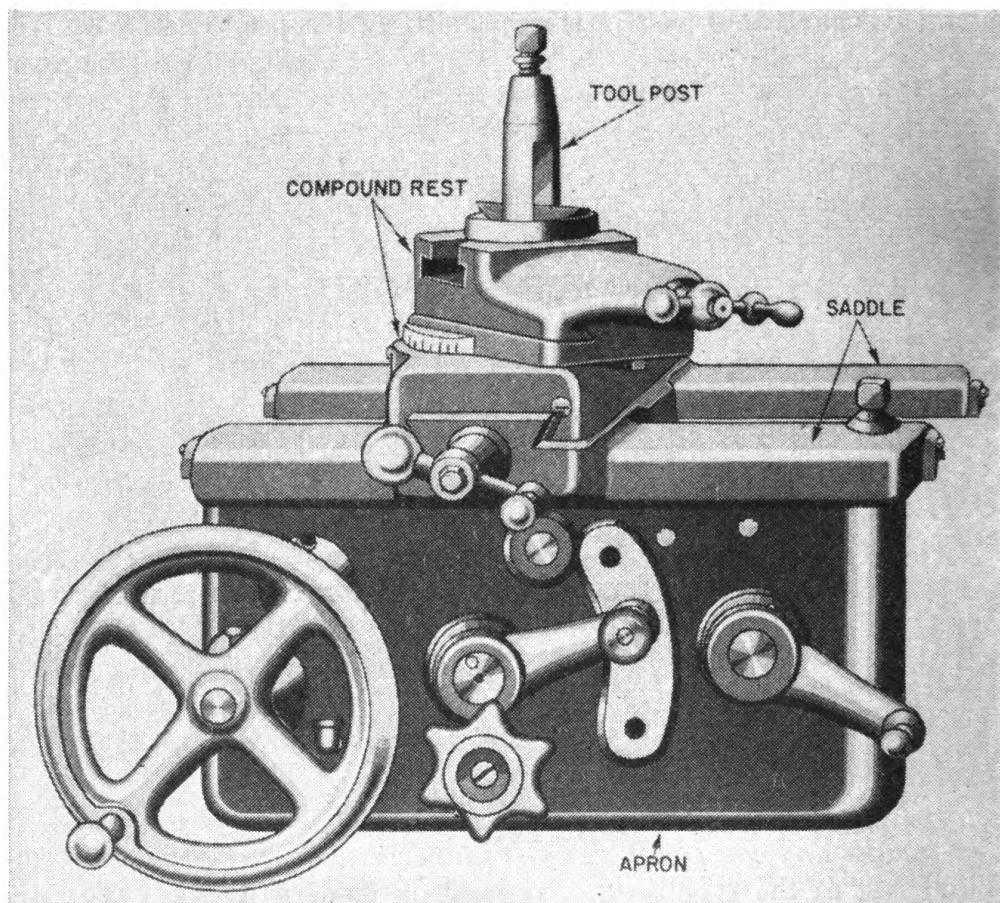


FIGURE 20.—Carriage.

operator be thoroughly familiar with the various controls of the apron, and understand how they are operated to secure the desired results.

(a) The carriage hand feed wheel moves the lathe carriage longitudinally along the ways.

(b) The apron feed clutch engages and disengages both the automatic longitudinal feed and the automatic cross feed; to engage the clutch, turn the knob to the right (clockwise); to disengage it, turn it to the left (counterclockwise).

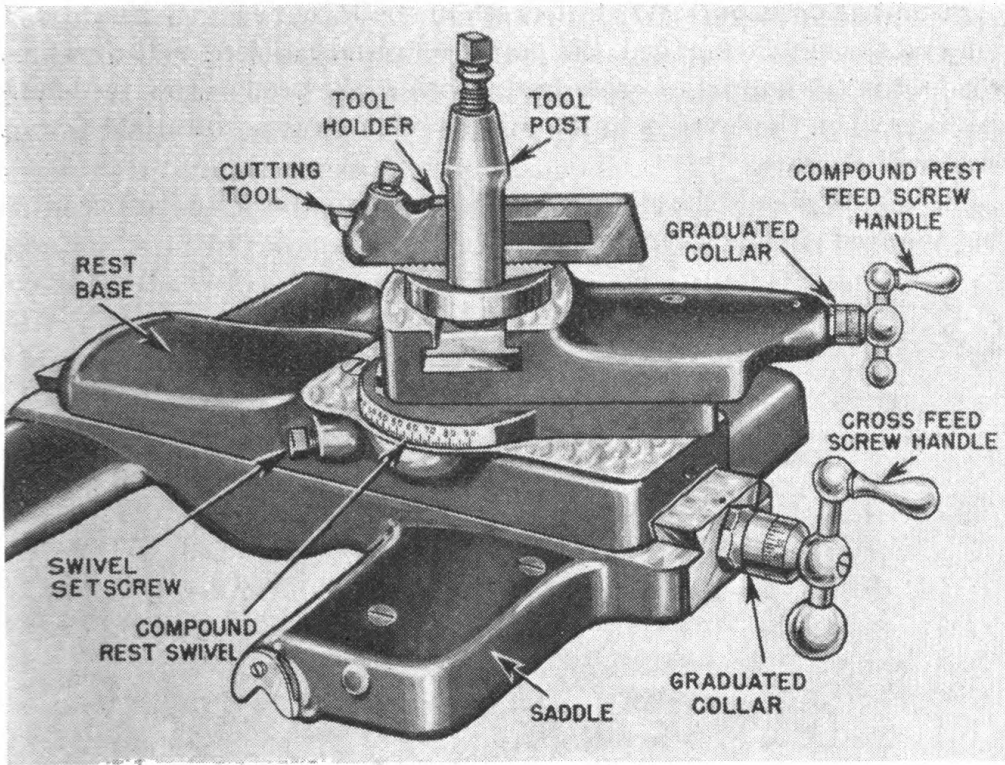


FIGURE 21.—Saddle and compound rest.

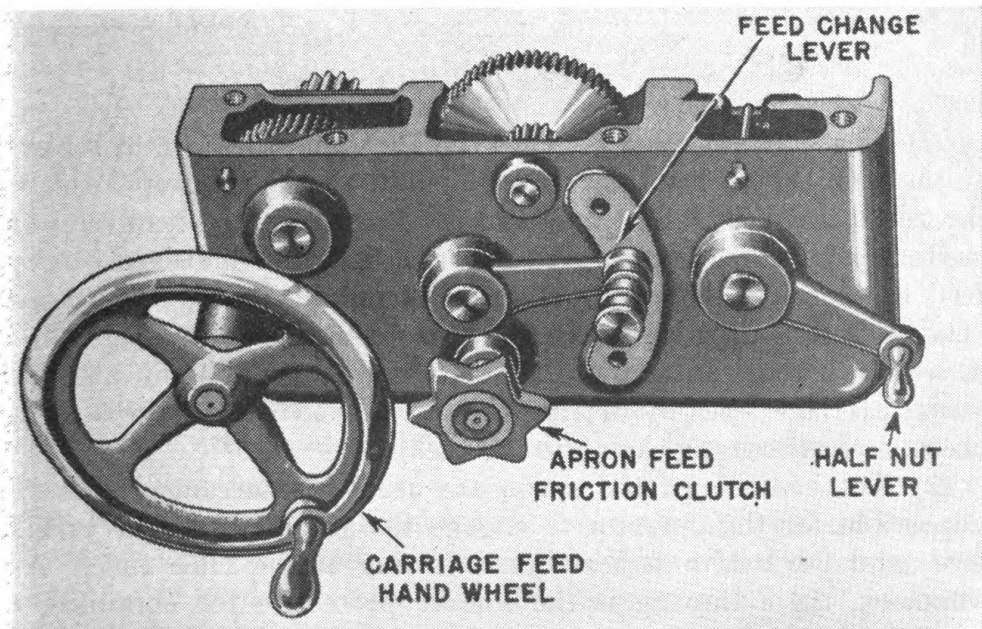


FIGURE 22.—Front of typical lathe apron.

(c) The feed change lever can be set in either the cross, neutral, or longitudinal position. When it is set in the top or longitudinal position and the clutch engaged, the power longitudinal feed will operate; when set in the bottom or cross feed position, the power cross feed will operate; when the lever is in the middle or neutral position, no power feeds will operate.

The clutch should be disengaged (turned to the left) before putting the feed change lever in neutral.

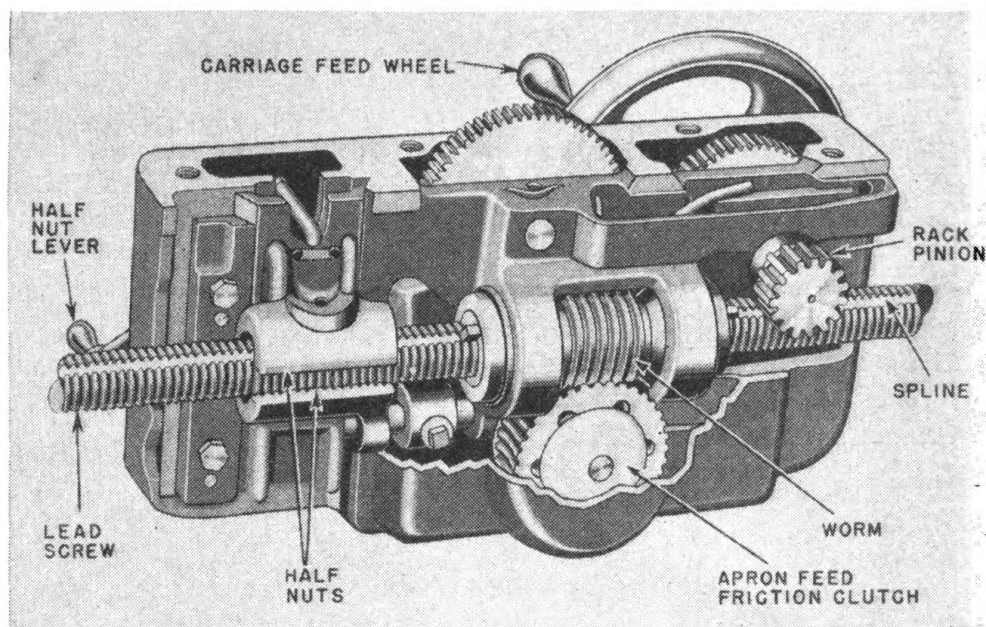


FIGURE 23.—Back of typical lathe apron.

(d) The half-nut lever engages the half nuts to the lead screw or disengages them; when they are engaged, the lathe is set up for screw-thread cutting. The half nuts and the threads of the lead screw are used *only* for thread cutting. Using the half nuts and lead screw to feed the carriage longitudinally for turning is not good practice; detailed instructions as to their use are contained in paragraph 20. A spline in the lead screw, not its threads, operates the automatic power carriage feeds by driving the worm shown in figure 23, which is connected through gearing to the rack.

(2) The controls of the apron are usually so arranged that it is impossible for the operator to engage the automatic power carriage feeds and the half nuts for thread cutting at the same time. As a summary, the following is the correct procedure for obtaining the carriage and feed movements available on most lathes:

(a) To move the carriage and the cross feed by hand, disengage the clutch by turning the clutch knob to the left; put the apron feed-change lever in neutral position; see that the half-nut lever is in the down or disengaged position; the carriage can then be moved longitudinally by turning the carriage hand feed wheel, and the cross feed by turning the cross feed screw handle.

(b) To engage the power longitudinal feed, set the apron feed change lever in the up or longitudinal position; engage the clutch by turning the clutch knob to the right; see that the half-nut lever is in the down or disengaged position. The carriage will then move longitudinally automatically.

(c) To engage the power cross feed, proceed as for engaging the power longitudinal feed, but put the apron feed change lever in the lowest or cross feed position.

(d) To cut threads, disengage the clutch by turning the clutch knob to the left; have the apron feed change lever in the middle or neutral position; then engage the half nuts to the lead screw by putting the half-nut lever in the up or engaged position.

i. Lead screw.—The lead screw of the lathe (figs. 16 and 23) is rotated by change gears mounted at the left end of the headstock, and drives the power carriage feeds by engaging the worm in the apron in a spline, and regulates the carriage travel for thread cutting by means of its own threads turning in the half nuts. Detailed instructions as to the use of the change gears, lead screw, and half nuts for thread cutting are given in paragraph 20. At present only the number of threads per inch of the lead screw of any lathe being operated need be remembered.

13. Cutting tools.—*a. General characteristics.*—To machine metal efficiently and accurately, cutting tools having keen, well supported cutting edges, ground for the particular metal being machined and the type of cut wanted must be used. Lathe tools or cutter bits are available in a number of materials, the most common of which are as follows:

(1) High-carbon, or tool steel high in carbon content, when correctly heated and quenched, hardens to a very high degree. However, this hardness is drawn at comparatively low heat, so when using carbon-steel cutting tools or bits take constant care to avoid overheating or “bluing” them.

(2) High-speed steel, of which most lathe cutting tools are made, is alloyed with tungsten and sometimes vanadium or molybdenum. Although not so hard as carbon steel when heat treated, high-speed steel retains its hardness at extremely high temperatures. Cutting tools

made of it can be used without damage at speeds and feeds which heat their cutting edges to a dull red.

(3) Stellite cutting tools are composed of chromium, cobalt, and sometimes tungsten. They are cast into form and cannot be forged as can carbon and high-speed steel tools. Under favorable conditions, cutting tools made of stellite will stand exceptionally fast speeds and heavy cuts.

(4) Tungsten carbide tipped cutting tools are used when maximum cutting speed is desired. Although expensive, they are highly efficient for machining cast iron, alloyed cast iron, copper, brass, bronze, aluminum, babbitt, and such abrasive nonmetallic materials as fiber, hard rubber, and bakelite. They are usually held in an open-side tool post for rigid support. Such tools require special grinding wheels for sharpening; they are too hard to be redressed on the shop abrasive wheel used for general tool grinding.

(5) Tantalum-carbide and titanium-carbide tipped tools are similar to those of tungsten carbide except that they are commonly used to machine steel.

b. Types of lathe tools.—Figure 24 shows the nine most commonly used types of lathe cutter bits and their applications.

(1) The left-hand turning tool is ground to be fed from left to right. The cutting edge is on the right side of the tool, and the top slopes down away from the cutting edge. The right side and front of the tool are ground with enough clearance to permit the cutting edge to advance when the feed is engaged without the heel of the tool rubbing against the work.

(2) The right-hand turning tool is just the opposite of the left-hand turning tool, being designed to cut when fed from right to left. It is ideal for taking roughing cuts and for general all-around machine work.

(3) The round-nosed turning tool is for general all-around machine work and is used for taking light roughing or finishing cuts. Usually the top is ground so that the tool may be fed from right to left, although it is sometimes ground flat on top so the feed may be in either direction.

(4) The left-hand facing tool is intended for facing on the left-hand side of the work (fig. 24). The direction of feed should be away from the axis of the work. The cutting edge is on the right-hand side of the tool, and the point is sharp to permit machining a square corner.

(5) The right-hand facing tool is just the opposite of the left-hand facing tool; it is intended to face the right end of the work or the right side of a shoulder.

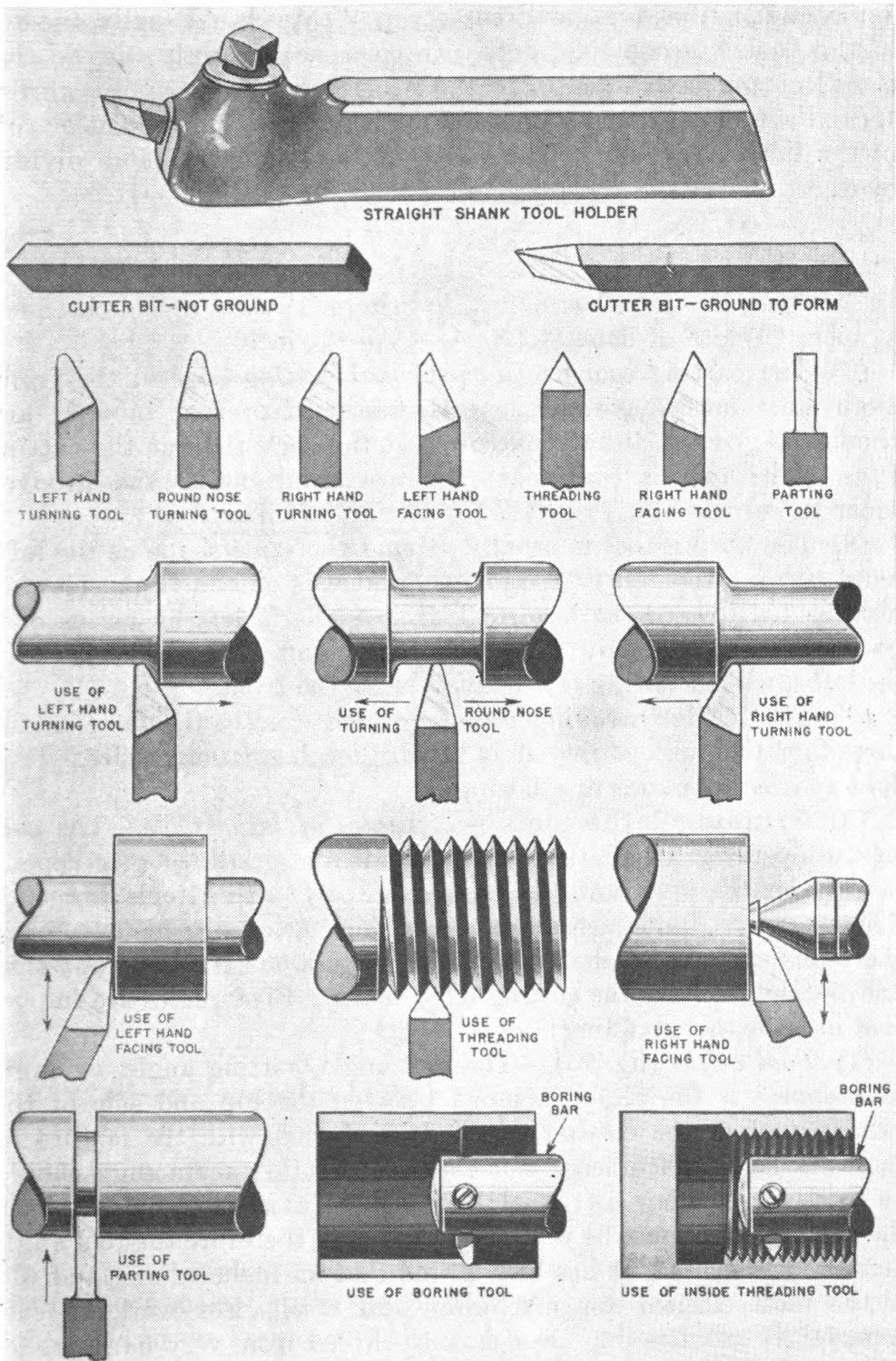


FIGURE 24.—Commonly used lathe tools and their applications.

(6) The threading tool has its point ground to a 60° cutting angle and in this form will cut sharp V-threads. Usually the top of this tool is ground flat and with clearance on both sides so that it will cut on both sides. For the American National screw thread form, the tool is ground with a flat at the point to cut the flat or root of the thread (fig. 66). The width of the flat is the pitch divided by 8:

$$\text{Flat} = \frac{\text{Pitch}}{8}$$

Example: A screw has 8 threads per inch; then the pitch equals $\frac{1}{8}$ inch. Width of flat = $\frac{1}{8} \div 8 = \frac{1}{8} \times \frac{1}{8} = \frac{1}{64}$ inch.

(7) The parting tool has its principal cutting edge at the front. Both sides must have sufficient clearance to prevent binding and should be ground slightly narrower at the back than at the cutting edge. The tool is convenient for machining necks and grooves, squaring corners, etc., as well as for cutting-off operations.

(8) The boring tool is usually ground the same shape as the left-hand turning tool, so that its cutting edge is on the front side and may be fed toward the headstock from right to left by means of a boring bar. Ordinarily a turning tool points toward the back of the lathe, while a boring tool points toward the front.

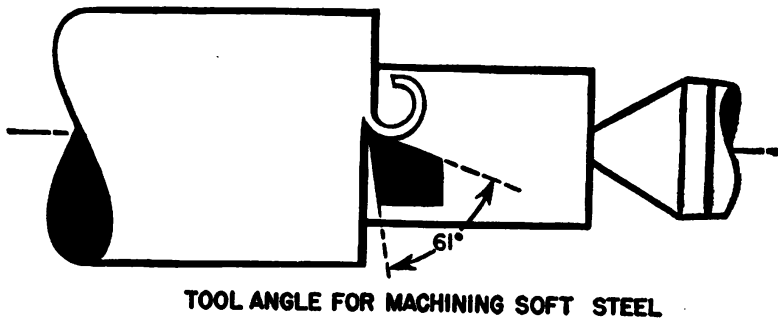
(9) The inside threading tool is ground exactly the same as the threading tool, except that it is usually much smaller in size. It is held and fed by means of a boring bar.

14. Grinding lathe tools.—*a. Angles of lathe tools.*—The successful operation of a lathe, and the grade of work done on it depend largely on the operator's ability to sharpen cutting tools properly. There are two fundamental objects of such grinding: first, to grind the tool's surfaces so that the cutting edge alone touches the work; and, second, to have the cutting edge sharp. Five angles are important in lathe tool grinding:

(1) *Tool angle* (fig. 25).—The tool angle, cutting angle, or angle of keenness is the angle included between the top and side of the cutting tool at the cutting edge. It is varied with the texture of the work being machined. For example, a rather acute angle should be used for turning soft steel; to machine hard steel or cast iron, the cutting edge must be well supported, and therefore the tool angle must be less acute. It has been found that an included angle of 61° is the most efficient for machining soft steel. For work on cast iron, 71° is satisfactory. For cutting chilled iron, very hard grades of cast iron, hard steel, bronze, etc., the tool angle may be as great as 85°.

THE MACHINIST

(2) *Clearance angles* (fig. 26).—Only the cutting edge of the tool must touch the work. It is necessary, therefore, that the surfaces of the tool next to the cutting edge be ground off at some angle which will keep them from rubbing on it. The angle which the side of a



TOOL ANGLE FOR MACHINING CAST IRON

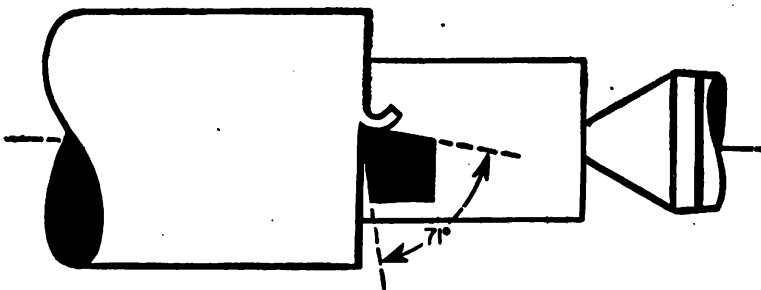


FIGURE 25.—Cutting angle of lathe tools.

tool makes with the vertical is called the side clearance angle. The angle that the front of the tool makes with the vertical is called the front clearance angle. With the tool in cutting position, the clearances must be not less than 3° and in most cases not more than 10°.

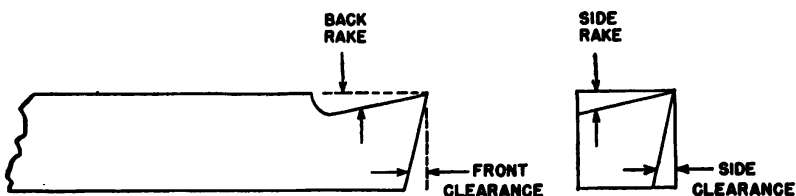


FIGURE 26.—Clearance and rake angles of lathe tools.

(3) *Rake angles* (fig. 26).—The angle which the upper side of a cutting tool makes with the horizontal is called the rake angle. Rake adds to the keenness of the tool and facilitates the removal of chips. If the slant is away from the work, it is a back rake angle; if in the direction of the axis of the work, it is a side rake angle;

if the slant of the back rake is down from the work, it is known as positive rake; if up from the work it is known as negative rake. Cutting tools may be ground with either back or side rake or a combination of both.

b. Grinding procedure.—(1) Before attempting to grind lathe tools, review paragraph 36, TM 10-590, for instruction in the use of power-driven bench grinders. The most satisfactory equipment for grinding lathe tools is a bench grinder with two 7-inch vitrified wheels, 36-grain for rough grinding and 60-grain for finish grinding. The wheel should be run at a surface speed of about 5,000 fpm and should have close fitting guards and tool rests. A small container of water should be at hand for cooling carbon-steel tools at intervals. High-speed steel tools should never be dipped in water when hot; this will crack the tool and crumble the cutting edge. Carbon-steel tools can also be ground on a wet wheel or an ordinary water-drip grindstone; high-speed steel tools are always ground dry. A simple way to tell whether a tool is carbon steel or high-speed steel is to grind the end and watch the sparks; the wheel will throw light-colored sparks from carbon steel, and dark red sparks from high-speed steel.

(2) Remove the cutter bit from the holder before sharpening. After tools have been ground on the abrasive wheels, they will produce better work and have longer life if the cutting edges are honed with an oilstone.

(3) Figure 27 shows the steps involved in grinding a round-nosed turning tool to be fed from right to left for general machine work. First grind the left side of the tool, holding it against the wheel at the correct angle to form the side clearance as shown in step 1. Use the coarse wheel to remove most of the metal, and finish on the fine wheel. Then grind the right side of the tool as shown in step 2. Do not remove any more metal than is necessary from this side as it requires no clearance and the more metal left on the bit the more heat it will absorb. Next, grind the radius or rounding on the end of the tool by holding it against the wheel and turning it from side to side as shown in step 3. Be careful to hold it at the correct angle. Then grind the front clearance (fig. 26) by holding the tool against the side of the wheel as shown in step 4 (fig. 27). Finally grind the top of the tool as shown in step 5, holding it at the right angle to obtain the necessary rake. The arrows show the correct direction of revolution of the wheel for each step.

c. Tool holders.—Cutter bits are generally made from standard sizes of bar stock to fit into a tool holder, which in turn is fastened

in the tool post of the lathe. The tool holder (fig. 24) will hold all the tools shown in the same illustration except the parting tool, the boring tool, and the inside threading tool. These require special holders which will be described. The holder (fig. 24) is called a straight shank tool holder; other types commonly used are right- and left-hand offset tool holders (fig. 28). They are generally used for holding cutter bits to perform facing operations, some counterboring operations, or generally when a straight shank tool holder is not practical.

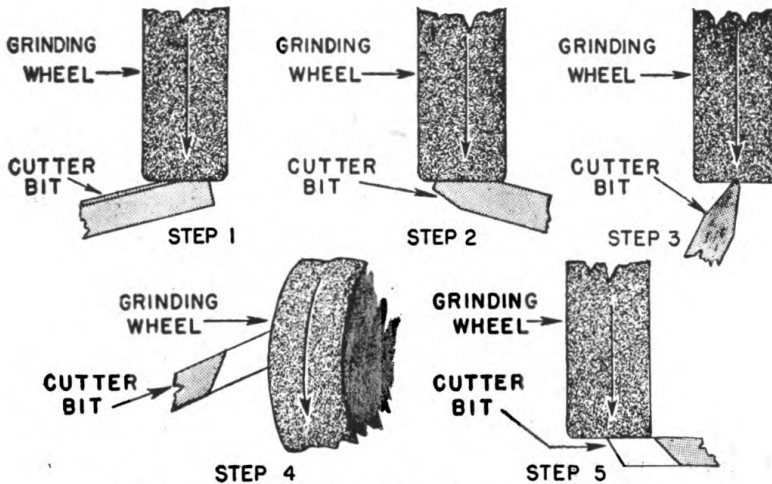


FIGURE 27.—Grinding a round-nosed turning tool.

d. Setting lathe tools.—(1) The cutting tool shank or, more commonly, the tool holder is held to the lathe carriage by a tool post. The type generally used is shown in detail in figure 29. The post consists of a piece with a slotted hole through its center for the tool holder. A collar slips over the post and rests on the compound rest; this collar may be beveled to provide vertical adjustment for the tool point. However, the most common practice is for it to be flat, and the vertical adjustment of the tool point is made by means of a rocker base (a metal piece in the shape of a segment of a circle, knurled on its flat edge) placed between the tool shank or holder and the collar. The tool post has a flange at the bottom which fits loosely into a slot in the compound rest. When the beveled collar has been turned, or the rocker base adjusted, to give the desired elevation to the tool point, the whole assembly can be clamped firmly into position for work by tightening the tool post screw as close to the tool post as possible; if it overhangs too far, the tool will spring under the pressure of the work, and chatter will result.

(2) For some classes of work, and for holding special types of cutting tools, an open side tool post (fig. 30) is often used to provide exceptionally rigid support for the cutting tool.

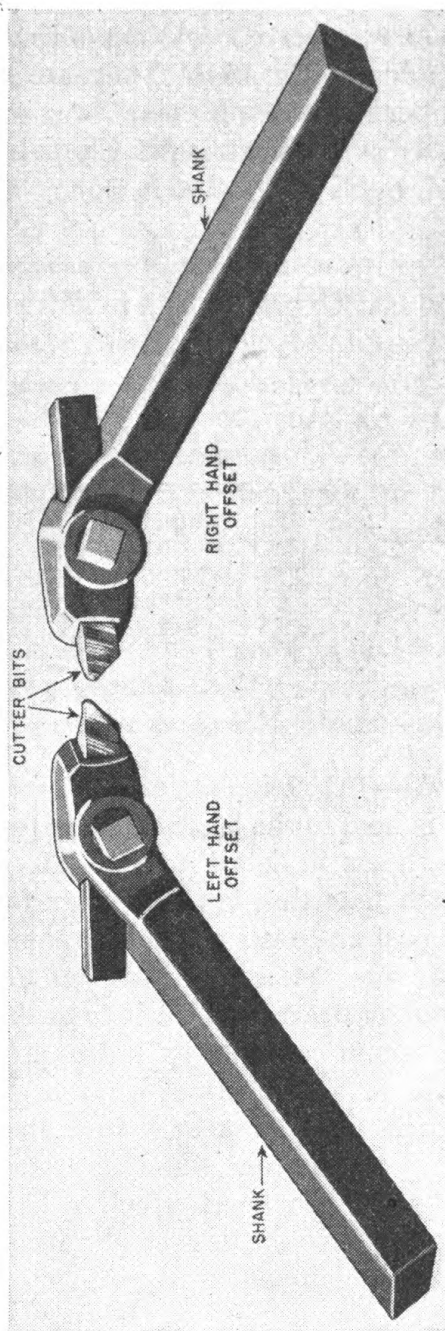


FIGURE 28.—Offset tool holders.

(3) For ordinary straight turning, the cutting edge of the tool should be about 5° , or $\frac{3}{64}$ inch per inch of diameter of the work, above a line between the lathe centers as shown in figure 31. If the

THE MACHINIST

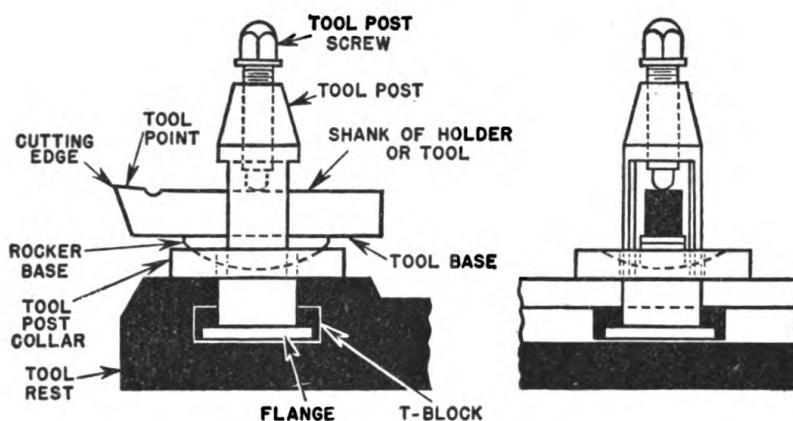


FIGURE 29.—Tool post assembly.

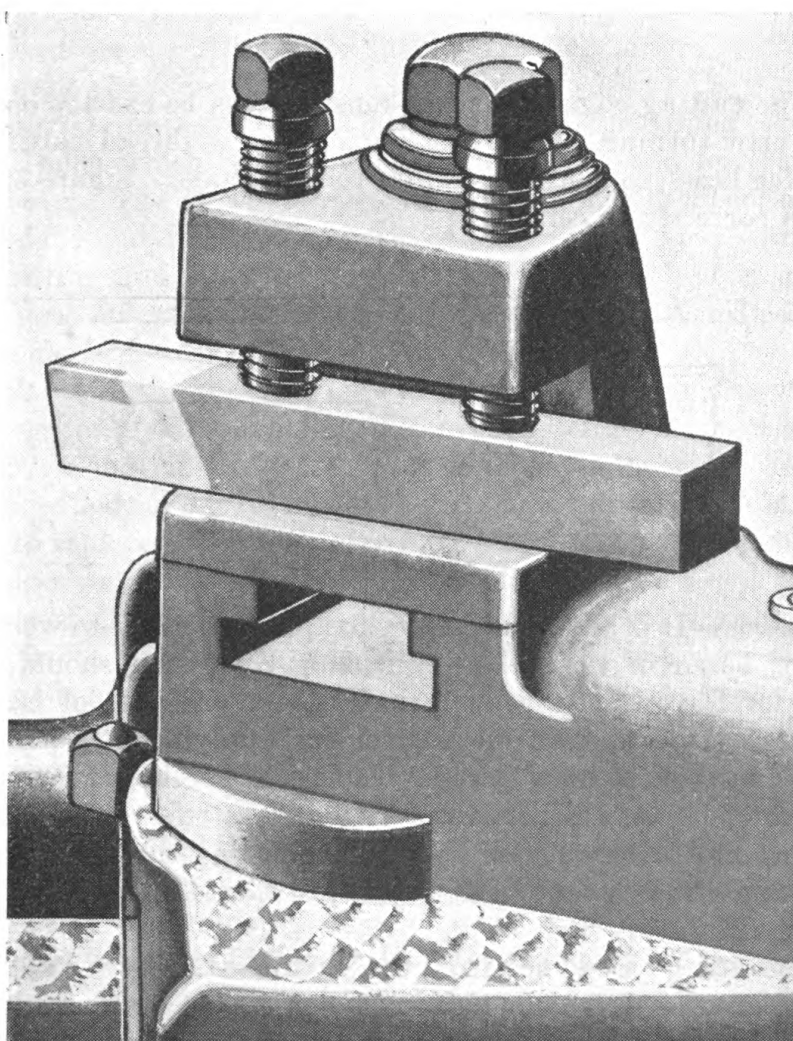


FIGURE 30.—Open side tool post.

tool is set too high, it will not have enough clearance and the work will rub against it and cause overheating and a rough cut. If set too low, the tool will not have enough rake, and the work will be scraped rather than cut. The strain on the tool will, moreover, be along its line of least resistance and may break it.

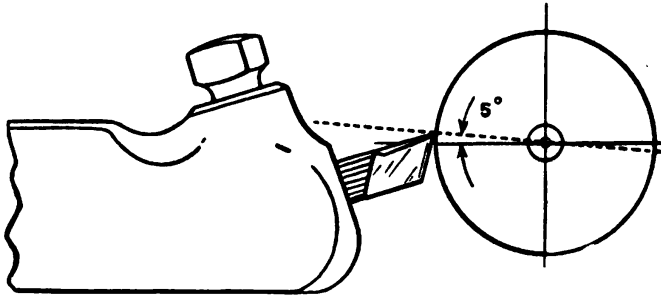


FIGURE 31.—Tool set 5° above center.

(4) The cutting edge of the tool must always be exactly on center for all taper turning and boring, facing, screw thread cutting, and for turning brass, copper, and similar tough metals. Figure 32 shows a tool set correctly on center.

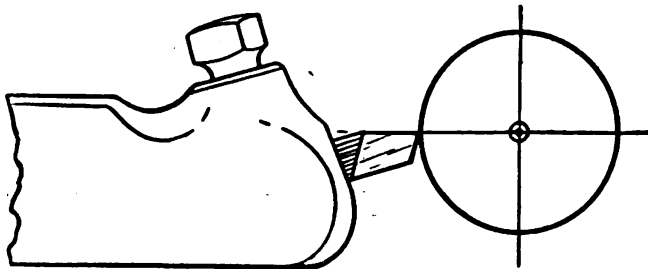


FIGURE 32.—Tool set on center for special work.

e. Cooling.—It is always necessary to apply a coolant to work being turned or bored in the lathe. A cutting compound should *always* be used for thread cutting operations except on brass or cast iron. Some large lathes have a mechanism for applying a coolant automatically by means of a pump. Table IX specifies recommended cutting compounds which have been found satisfactory for various operations on different materials. When cutting threads, many machinists apply the coolant to the work just ahead of the cutting tool with a small brush.

15. Leveling and setting up lathe.—*a. Floor lathes.*—(1) Unless a lathe is level, no machinist can do accurate work on it. The machine must be on a solid foundation, preferably a concrete floor; a wood floor should be braced if necessary to prevent sagging

and vibration. The usual method of leveling a lathe is to put tapered shims or wedges of hardwood or metal under each leg. If each wedge is hit lightly with a hammer, it will slightly raise the leg. To test the level of a lathe, use a precision level at least 12 inches long and sensitive enough to show a distinct movement of the bubble when a 0.003-inch shim is placed under one end of it. (An ordinary carpenter's or combination set level is not sensitive enough for this operation.) Place the level across the bed, as close to the headstock as possible, and drive the wedges under the headstock legs until the bubble is in the center of the tube. Follow the same procedure with the level as near the tailstock end of the bed as possible. Repeating this operation several times will bring both ends of the bed to a perfect level.

(2) It is also necessary to level the lathe lengthwise. The lathe should be fastened to the floor firmly with lag screws, and its level tested again after fastening.

b. Bench lathes.—A bench lathe should be mounted on a substantial bench which will provide rigid support, and leveled as outlined above. The bench top should be of 2-inch lumber, about 28 inches high, and the whole bench bolted to the floor to prevent it from shifting and throwing the lathe out of level. If at any time a lathe does not bore a straight hole it has probably shifted and its level should be readjusted.

c. Belt lacing and tension.—(1) Since the driving power of a lathe is generally transmitted to it by belts either from a lineshaft or from an individual motor through cone pulleys, the machinist is often called upon to measure, cut, and splice, or join leather belting to replace that which may have worn out or become damaged. The easiest way to determine the required length of a belt is to pass a flexible steel tape over the pulleys to be connected and read the length direct. If such a tape is not available, the following procedure will give the measurement wanted: Add the diameters of the pulleys in inches, and multiply the sum by 1.57; then add to this product twice the distance in inches between centers. The length of small belts, such as are commonly used in lathe headstocks, can be found by passing the piece to be used over the pulleys and straining it by hand, being sure that mating steps on the cone pulleys are selected.

(2) The three commonly used methods of joining leather belting are lacing, gluing, or joining with wire belt hooks. Figure 33 shows the method of joining a belt by lacing. The correct procedure is first to cut the belt to the desired length and trim the ends square. Punch or drill holes just large enough for the lace in each end of

the belt as shown in the illustration. (Wide belts require more holes but are seldom used in lathe set-up work.) The thongs are made of gut or rawhide; if a round gut lace is used, cut straight grooves $\frac{1}{8}$ inch wide and $\frac{1}{16}$ inch deep on the smooth side of the belt from the holes to the ends. This will make the belt run smoothly over the pulleys. Start lacing from the grooved side of the belt through the center holes and pull both ends of the lace through evenly;

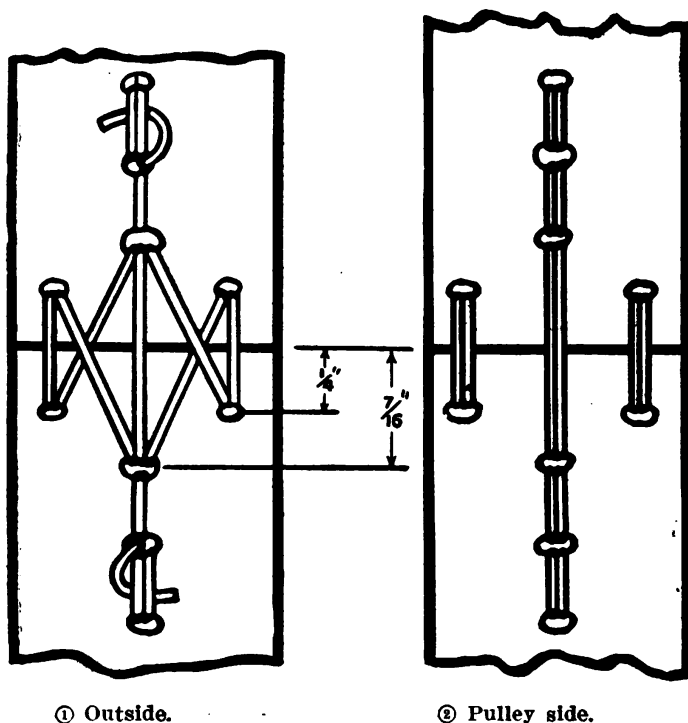


FIGURE 33.—Laced belt joint.

work out to both edges, then back to the center, and fasten the ends of the lace as shown in figure 33. Always remember that the *smooth* side of a leather belt is the side that should run on pulleys.

(3) Perhaps the best all-around method of joining belts is to splice them by gluing (fig. 34). To make a good glued splice, taper the ends of the belt uniformly (being sure you have provided enough length for the taper when figuring the length required); use any good belt glue and keep the splice under pressure with clamps until the glue sets.

(4) Wire belt hooks (fig. 35) are applied with a machine made for the purpose; they save time and are easy to use but are not suitable for belts which are to be shifted from one pulley to another while in

motion. Motor-driven lathes are provided with an attachment for adjusting the tension of the cone pulley belt; it should be set so that the belt is just tight enough to transmit the power required. Belts that are too tight wear out more quickly than they should, waste power, and throw an unnecessary strain on the pulley bearings. New belts will stretch to some extent. Belts and pulleys should always be kept dry and clean, and free from oil or grease.

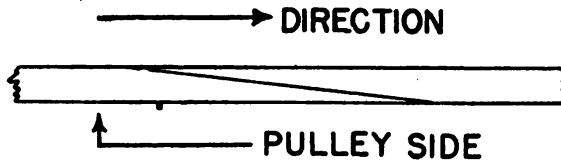


FIGURE 34.—Glued belt splice.

d. Shifting belts.—To shift the belt on a countershaft-driven lathe, the beginner should always stop the lathe, and shift by hand by slipping and pulling the belt to the desired position. Motor-driven lathes should always be stopped before the cone pulley belt is shifted; they are provided with a lever which will loosen the belt to make the shift easy and quick.

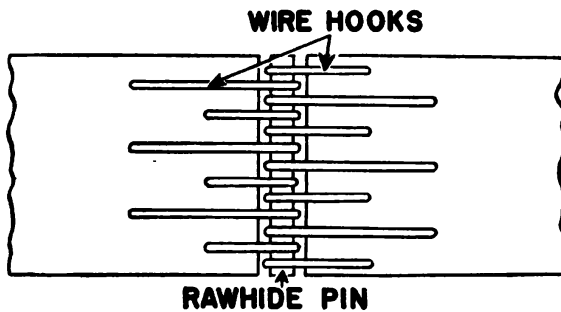


FIGURE 35.—Wire belt joint.

e. Lubrication.—(1) The lubrication procedure in (2) below for any lathe will insure its giving good service over a long period of time; if neglected, the machine will produce inaccurate work and probably develop serious break-downs.

(2) Oil every bearing on the lathe as directed by the manufacturer's instruction chart. Do not attempt to oil a lathe while it is running. Use a good grade of machine oil, *not* automobile engine oil. Oil a new lathe twice daily for the first week or so; once daily thereafter. Do not run a new lathe at a faster spindle speed than 500 rpm until it is thoroughly broken in. It is good practice always to oil the bearings of a lathe in the same order so that no oil holes will be missed. A few drops of oil in each hole is sufficient; excess oil around the bear-

ings should be wiped off with a rag or piece of waste. Oil the counter-shaft each time the lathe is oiled; if the lathe is motor driven, oil the motor bearings once a week. Do not allow oil, dirt, chips, or rust to collect anywhere on the lathe.

16. Plain turning (work between centers).—*a. Locating centers.*—(1) The usual method of holding work for plain turning in a lathe is to mount it between pointed centers in the headstock and tailstock. Work should be set up in this manner whenever it is practical to do so, especially if it is heavy or if deep cuts are to be taken, because the lathe centers support it at both ends. Work set up in this manner must have conical cavities in the ends to fit the lathe centers. The process of drilling and countersinking these cavities is called centering, and consists of first locating their position, and then drilling and countersinking them to the correct size and depth.

(2) The centers must be located so that the entire diameter of the turned job will finish to size, and, for good work, the chip taken should be of practically uniform depth as the work rotates. To obtain these two conditions, the centers must be drilled as close as possible to the actual center of the piece to be turned. Several satisfactory procedures for so locating them on the ends of bar stock are in common use.

(a) *Hermaphrodite caliper method.*—Set a pair of hermaphrodite calipers to approximately one-half the diameter of the work and scribe four short arcs as shown in figure 36. The arcs will inclose the center.

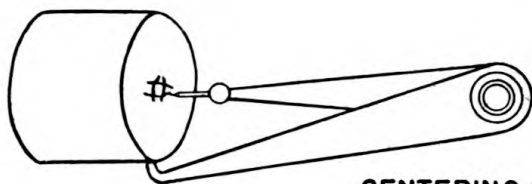
(b) *Center head method.*—Hold the center head of a combination set firmly against the work (fig. 36) and scribe a line close to the blade; give the bar a quarter turn and scribe a similar intersecting line. The point where the lines intersect will be the center.

(c) *Bell center punch method.*—If a bell centering punch (fig. 36) is available, place it over the end of the stock and strike the plunger a sharp blow with a hammer. The bell automatically locates the punch mark in the center of the work. In using this method, see that the stock is cut off square at the end and that it is free from burs. The plunger must be held in line with the axis of the stock, or the resulting mark will be off center.

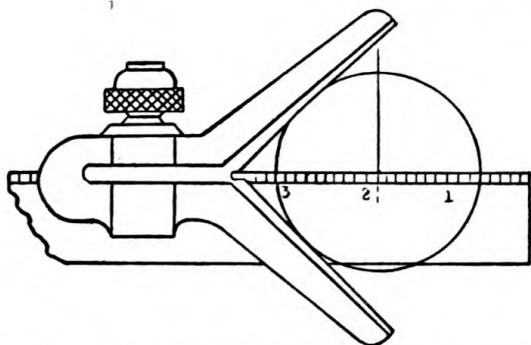
(d) *Divider method.*—With the work on a flat surface, set the dividers to approximately one-half the diameter of the work and scribe four lines across each end as shown in figure 36. The center will be within the small square thus formed. No definite rules can be given for centering irregular work. Special centering jobs de-

mand common sense and good judgment, and some laying-out experience, if the centers are to be located correctly.

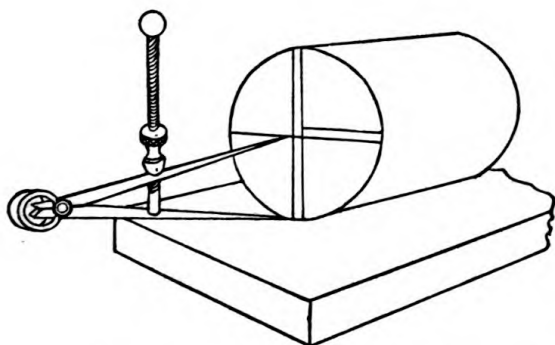
b. Testing location of centers.—When the centers have been located, test them for accuracy before drilling and countersinking them. The



**CENTERING WITH
HERMAPHRODITE CALIPERS**

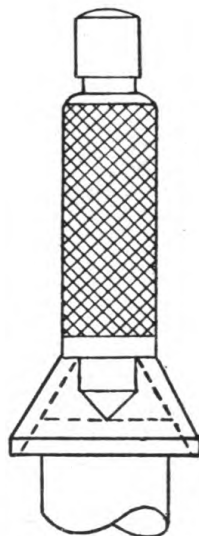


USE OF CENTER HEAD TO LOCATE CENTER



LOCATING CENTER WITH DIVIDERS

FIGURE 36.—Locating centers on bar stock.



BELL CENTER PUNCH

usual procedure is to indent them with a center punch deeply enough so that the work can be held lightly between the lathe centers as shown in figure 37. By holding a piece of chalk stationary against the work (preferably on a rest), and spinning the work as shown in the illustration, any high spots can be clearly marked at either end of the piece and determined whether the centers have been located with

sufficient accuracy. Remember that absolute accuracy is not required nor, in fact, likely to be attained; but the more accurately located the centers are, the better the job will be. It is often necessary to change the location of one or both of the center punch marks before drilling. A satisfactory method of doing so is to hold the work firmly in a vise, and by means of a center punch held at an angle, drive the center to be moved in the desired direction.

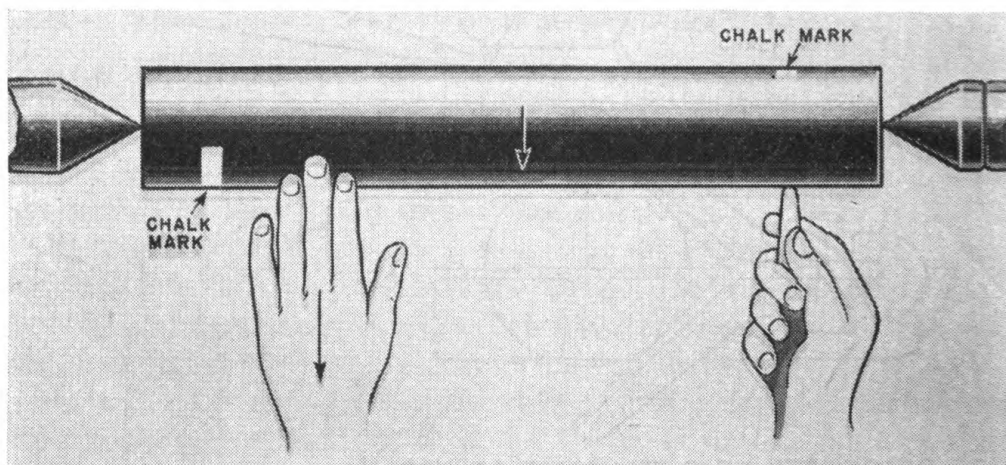
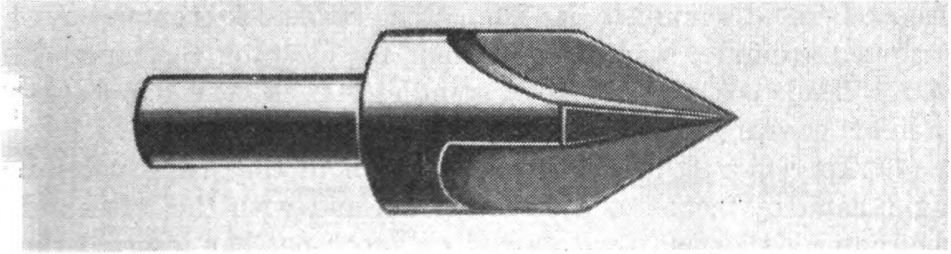


FIGURE 37.—Method of testing location of centers.

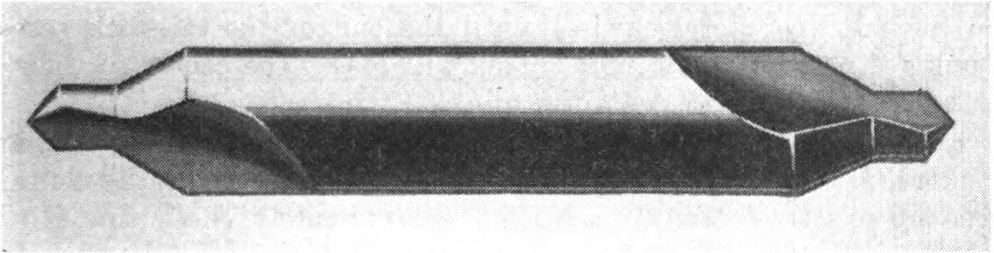
c. Drilling center holes.—(1) When the centers have been properly located on the ends of the work, the next step is to drill and countersink them the right size and depth to fit the lathe centers. This can be done either by means of a small twist drill followed by a 60° countersink or, more commonly, by means of a combination drill and countersink (fig. 38). It is very important that the center holes be drilled and countersunk so that they fit the lathe centers exactly. Poorly drilled center holes are among the most common causes of unsatisfactory lathe work. Lathe centers are tapered to an included angle of 60° and the center holes in the work must taper at the same angle, with some clearance remaining at the bottom of the hole. Figure 39 shows a correctly drilled and countersunk center hole, and the way it should fit the lathe center. If the hole is too shallow or too deep, or if no clearance is provided at the bottom of the hole, the work cannot be machined accurately. Carelessly drilled and countersunk work centers do not give the lathe centers full bearing on the work, and as a consequence the work does not run true, and the lathe centers are subject to unnecessary wear. Table X lists the best sizes of center holes for commonly used diameters of bar stock which can be drilled with generally available combination drills and counter-

sinks. The actual drilling operation can be performed either by a drill press (par. 9c) or by drilling in the lathe itself (par. 19).

(2) Centering requires some care. The spindle speed should be about 600 rpm and the feed kept comparatively light to avoid any possibility of breaking the drill point. A broken point requires an



① 60° countersink.



② Combination drill and countersink.

FIGURE 38.

annealing operation to remove it, which may spoil the work. When drilling center holes, allow for the thickness of work metal that will be faced off at the end; if this allowance is overlooked, the hole may be too shallow after the facing operation is completed.

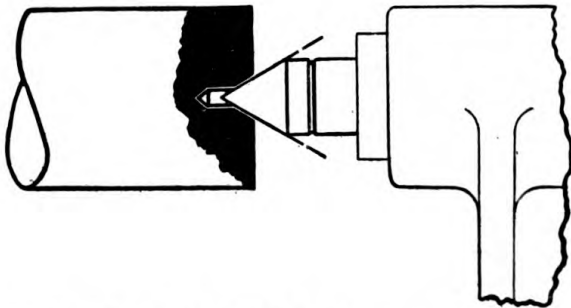


FIGURE 39.—Correctly drilled and countersunk work center.

d. Inserting and removing lathe centers.—The quality of work done on a lathe depends to a great extent on the condition of the lathe centers. Before mounting the centers in the headstock (live

center) or tailstock (dead center), always *thoroughly* clean them, as well as the tapered holes in the headstock and tailstock, and the spindle sleeve. A very small amount of dirt or a small chip on the center or in the spindle will cause the centers to run out of true. The tailstock or dead center, which does not revolve with the work as does the headstock or live center, is hardened and tempered, and grooved to distinguish it from the headstock center. A cloth wrapped around a stick is convenient for cleaning the tapered holes in the headstock and tailstock spindles. It is very dangerous ever to insert the finger into a revolving spindle.

(1) To remove the headstock center, hold the sharp point in the right hand by means of a cloth or rag, and with the left hand give the center a sharp tap with a rod or knock-out bar inserted through the hollow spindle.

(2) To remove the tailstock center, turn the tailstock handwheel to the left (counterclockwise) until the end of the tailstock screw bumps the end of the center. This will loosen the center so that it can easily be removed from the spindle.

e. Alining centers.—For plain turning on work set up between centers, it is absolutely essential for accuracy that the lathe centers line up exactly. Many machinists approximately check this alinement by eye by moving the tailstock close to the headstock (fig. 40). If the centers do not line up, they can be corrected by adjusting the position of the tailstock top by the adjusting screws in its base.

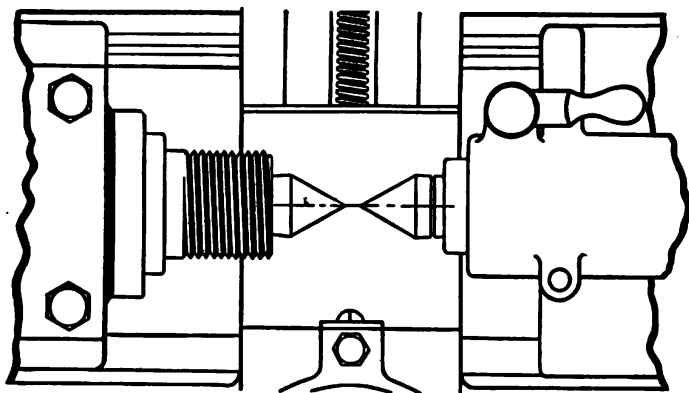


FIGURE 40.—Checking alinement of lathe centers by eye.

This procedure will approximately aline the centers; a more exact adjustment can be made after the first roughing cut is taken across the work; the method of doing so is explained in paragraph 16i.

f. Setting up work between centers.—(1) When the lathe centers have been properly inserted in place and approximately alined, and

the work correctly drilled for mounting, the next step is to mount the piece in the lathe, usually by means of a faceplate and lathe dog at the headstock end as shown in figure 41.

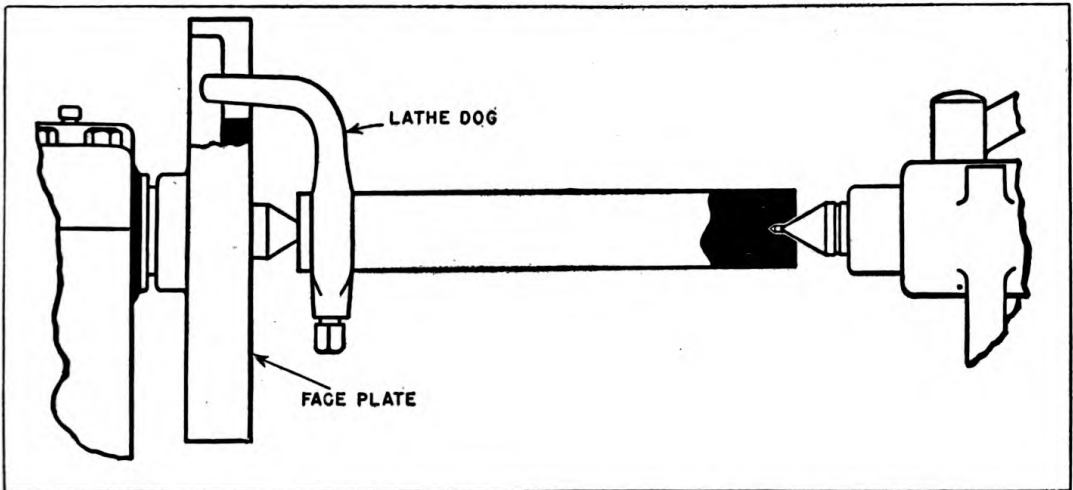
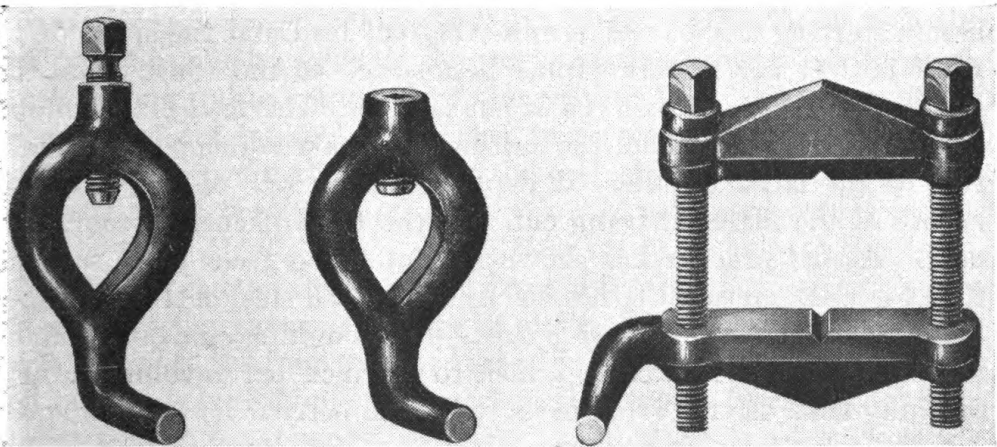


FIGURE 41.—Work correctly set up in lathe for plain turning.

(2) The purpose of the lathe dog is to provide a firm connection between the headstock spindle and the work, and thus drive the work at the same rpm as the spindle under the strain of cutting. Figure 42 shows three commonly used types of lathe dogs. The part



① Common.

② Safety.

③ Clamp.

FIGURE 42.—Lathe dogs.

of the dog which fits into a slot in the faceplate is called the tail, which may be either bent, as shown, or straight. Straight tail lathe dogs are driven by means of a stud attached to and projecting from the faceplate. The safety lathe dog has a headless setscrew to reduce

the danger of its catching in the operator's sleeve, which can cause a serious accident. The clamp lathe dog is used principally for driving rectangular work.

(3) Take particular care that the threads of the headstock spindle sleeve are clean before screwing the faceplate to them, and avoid damaging them in any way. The setscrew of the lathe dog should be turned tightly on the work; if the work is finished, it is good practice to place a split ring of some soft material such as brass between the setscrew and the work. Make sure that the tail of the lathe dog does not bind in the slot of the faceplate.

(4) Since the dead center (tailstock) does not revolve with the work as does the live center (headstock), it requires lubrication; a few drops of oil or white lead should be applied to the dead center before the work is set up, and the tailstock adjusted so that the dead center fits firmly, but not so firmly as to bind, into the cavity previously countersunk in the work. It is advisable to apply white lead to the dead center at intervals during the turning operation to reduce wear and prevent damage either to it or to the work. The heat generated during the turning operation causes the work to elongate which can cause unnecessary wear on the dead center. Lubrication and pressure upon the dead center must be checked frequently during the turning operation if accurate work is to be done without damaging the lathe.

g. Cutting feeds and speeds.—(1) When the work is set up and before starting any cut, determine the tool feed and the speed of the work in feet per minute (fpm) best suited to the work metal, the size of the lathe, and the type of cut needed. The feed is the amount of longitudinal movement the cutting tool makes during each revolution of the lathe spindle. It depends on the size of the lathe, the nature of the material being cut, and the type of cutting tool being used. Actual practice has shown the following feeds to be satisfactory for work on small lathes, using high-speed steel cutting tools:

(a) *Cast iron and malleable iron.*—For roughing cuts on diameters up to 8 inches, feed from $\frac{1}{32}$ inch to $\frac{1}{16}$ inch per revolution. For finishing cuts, use the same feeds with a 25 percent increase in speed.

(b) *Cast steel.*—For roughing cuts on diameters up to 8 inches, feed from 0.020 inch to 0.040 inch per revolution. For finishing cuts, decrease the feed and increase the speed from 25 to 50 percent.

(c) *Machine-steel forgings.*—For roughing cuts on diameters up to 8 inches, feed from 0.020 inch to 0.040 inch per revolution. For finishing cuts, decrease the feed from 20 to 30 percent and increase the speed from 30 to 50 percent.

(d) *Machine-steel bar stock*.—Feed from 0.020 inch to 0.040 inch if the stock reduction is not more than $\frac{3}{16}$ inch to $\frac{1}{4}$ inch. If the stock reduction is increased, reduce the feed to not more than 0.015 inch per revolution. For finishing cuts, increase the speed as much as 50 percent, but do not increase the feed.

(e) *Tool-steel forgings and steel alloys*.—For roughing cuts on diameters up to 8 inches, feed from 0.012 inch to 0.020 inch if the depth of cut is not more than $\frac{1}{4}$ inch. For finishing cuts, use the same feed but increase the speed 25 percent.

(f) *Yellow brass*.—Using a tool with negative rake for roughing cuts on diameters up to 8 inches, feed from 0.020 inch to 0.040 inch. For finishing cuts, use the same feed and increase the speed.

(g) *Composition brass*.—Slightly lighter feed than for yellow brass.

(h) *Bronzes*.—For roughing cuts on diameters up to 8 inches, feed from 0.020 inch to 0.040 inch. For finishing cuts, increase both feed and speed 25 percent.

(i) *Aluminum*.—Feeds vary because of differences in the composition of the alloy. Generally, 0.040 inch to 0.080 inch per revolution will be found satisfactory for roughing cuts on small diameters. Finishing cuts are possible at feeds from $\frac{1}{8}$ inch to $\frac{1}{2}$ inch per revolution, provided broad-nosed tools are used.

(2) A lathe equipped with change gears generally has an etched plate conveniently visible that specifies the gearing required to produce various automatic longitudinal and cross feeds in fractions of an inch per revolution. The feed should not be confused with the depth of cut, which regulates the reduction in diameter of the work for each longitudinal traverse of the cutting tool. For example, if the depth of cut into a bar 2 inches in diameter is $\frac{1}{16}$ inch, at the end of one longitudinal traverse of the tool, the diameter of the work would be reduced twice that amount, or $\frac{1}{8}$ inch, and would be $1\frac{7}{8}$ inches.

(3) The speed of the work is given as its surface speed in feet per minute, the same as for twist drills as previously explained. Table XI gives the recommended speed in fpm for turning commonly used materials with high-speed steel cutting tools. These speeds may be increased 25 to 50 percent if a cutting lubricant is used; if stellite or tungsten carbide tipped cutting tools are used, further increases are practicable. If too slow a cutting speed is used, much time will be lost; if the speed is too fast, the tool will become dull quickly.

(4) To find the number of rpm at which to run the lathe spindle to obtain a given cutting speed in fpm, multiply the given cutting

speed by 12, and divide this product by the circumference in inches of the work. (The circumference is the diameter multiplied by 3.1416.) For example, if you want to turn a 1-inch diameter shaft at a speed of 90 fpm, the calculation would be as follows:

$$\frac{90 \times 12}{3.1416 \times 1} = 343.77 \text{ rpm}$$

To eliminate the necessity of making frequent calculations, refer to table XII which gives the spindle speeds in rpm necessary to obtain various cutting speeds in fpm for work from 1 to 16 inches in diameter.

h. Facing.—The first operation usually performed on work in the lathe is facing or squaring the ends in order to obtain a uniform bearing for the centers, and often to bring the work to some specified length. The usual procedure is first to face the right end then the left end. The right-hand facing tool (fig. 24) is most satisfactory for facing the right end of the work, being so shaped that it will cut without interfering with the dead center of the lathe. For the first, or roughing cut, set the tool on center at a slight angle to the work surface (fig. 43) and begin the cut as close as possible to

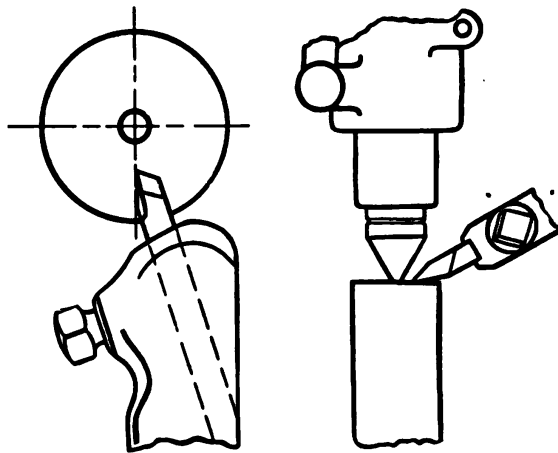


FIGURE 43.—Cutting tool set for right-hand facing.

the axis of the work and feed the tool *out* away from the dead center. Remove only enough metal to square the end over its entire area. Follow this cut with a finishing cut with the cutting edge of the tool set nearly flat against the work surface, removing a light, thin chip. In starting the facing operation, care is needed to

see that the cutting tool does not touch the dead center of the lathe. To face the left or headstock end of the work, the procedure is the same except that the left-hand facing tool (fig. 24) must be used.

i. *Turning*.—(1) As a rule, work is turned down to some predetermined diameter which is smaller than that of the rough stock. The usual procedure is to take heavy roughing cuts until the work is within about $\frac{1}{32}$ to $\frac{1}{64}$ inch of the diameter desired and remove the remaining metal by means of one or more very light finishing cuts. Whenever possible, feed the cutting tool from right to left toward the headstock to press the work against the live center, using a right-hand turning tool (fig. 24) which is ideal for the purpose. If circumstances demand that the feed be toward the tailstock from left to right, a left-hand tool must be used instead of the right-hand tool.

(2) Paragraph 16e describes the method of alining the lathe centers approximately by eye. Exact alinement can be obtained as follows: After taking the first roughing cut along the whole length of the work, measure its diameter at a point close to the dead center, and without changing the setting of the calipers used compare this measurement with its diameter at a point close to the live center. Unless the measurements are the same, the lathe centers are not exactly in alinement, and the error should be corrected at once by adjusting the position of the dead center by means of the adjusting screws in the tailstock. If the work is larger at the tailstock end, move the dead center back; if larger at the headstock end, move the dead center forward. These adjustments should generally be very slight, since any movement of the dead center will change the diameter of the work by twice the distance it is moved. When measuring the work, remember that it is extremely bad practice to caliper any work while the lathe is in motion.

(3) When the lathe centers have been alined, continue taking roughing cuts as deep as the lathe and cutting tool will allow. After each traverse of the tool, check the diameter of the work, until it has been reduced to within about $\frac{1}{32}$ to $\frac{1}{64}$ inch of the finished size desired. At that point, reduce the depth of cut and increase the spindle speed and longitudinal feed, and take very light finishing cuts until the work is turned to the diameter wanted. The round-nosed turning tool (fig. 24), is very efficient for finishing cuts. Finishing cuts should be measured after each traverse of the tool with a micrometer caliper rather than ordinary calipers. *Remember that if the work is turned smaller than the finished size desired, it cannot be made larger and is consequently ruined.*

j. Machining to a shoulder.—(1) It is frequently necessary to turn a piece of work so that it will have two or more diameters in its length. For example, a bar 12 inches long might be 3 inches in diameter for half its length, and $3\frac{1}{2}$ inches in diameter for the other half. It would therefore have an abrupt step or shoulder $\frac{1}{4}$ inch high 6 inches from the small end. This shoulder may be machined so that it forms a sharp corner with the small diameter, or a fillet may be formed so that the corner is rounded slightly instead of being left square. The important step in machining a shoulder is to locate it correctly on the work at the beginning. Several methods are satisfactory, depending on the degree of accuracy that the job demands. In any method, the first step is to turn the entire piece to the largest diameter desired (in the example being used, $3\frac{1}{2}$ inches). The location of the shoulder can then be marked by chalking the work and scribing it 6 inches from the end with a pair of hermaphrodite calipers as shown in figure 44. Then continue to turn down the end

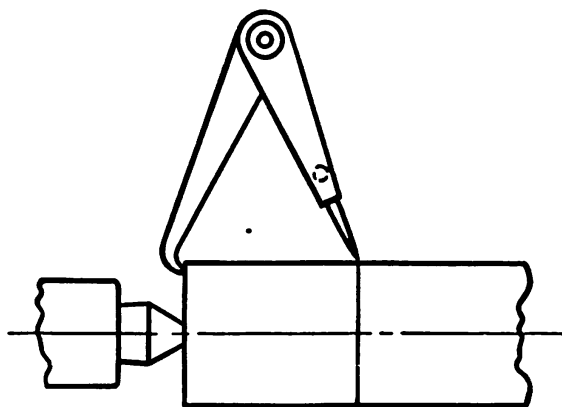


FIGURE 44.—Locating a shoulder with Hermaphrodite calipers.

of the work which is to have a 3-inch diameter, stopping the traverse of the cutting tool about $\frac{1}{32}$ inch before it reaches the scribed line. This $\frac{1}{32}$ inch is important, because it leaves sufficient metal on the shoulder to provide for a facing operation after the smaller part of the work has been turned to size.

(2) When the diameters have been turned, the final operation is to face the shoulder with a facing tool, feeding from the circumference of the work toward the center. If the shoulder is to have a fillet or small female radius at its base, this fillet can be formed by using a round-nosed turning tool set at an angle as shown in figure 45. If a fillet is wanted, an extra $\frac{1}{64}$ inch of metal must be left on the shoulder to provide for it. In cases where the shoulder must be accurately located, many operators first take a cut where the

shoulder is to be with a parting tool as shown in figure 46. If a shoulder is to carry a bearing it is often left undercut or recessed as shown in figure 47.

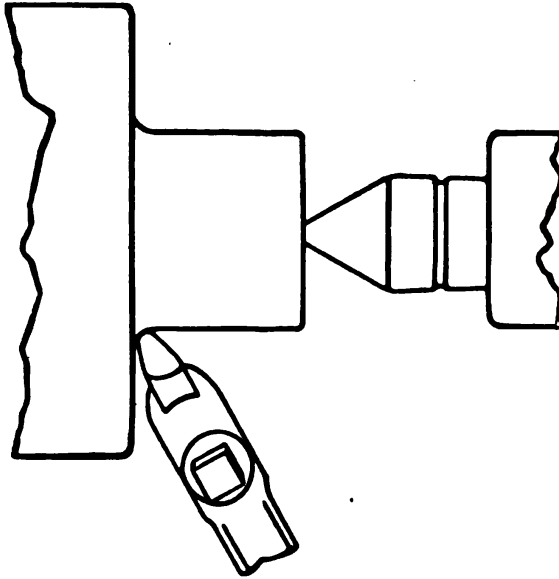


FIGURE 45.—Forming fillet with round-nosed tool.

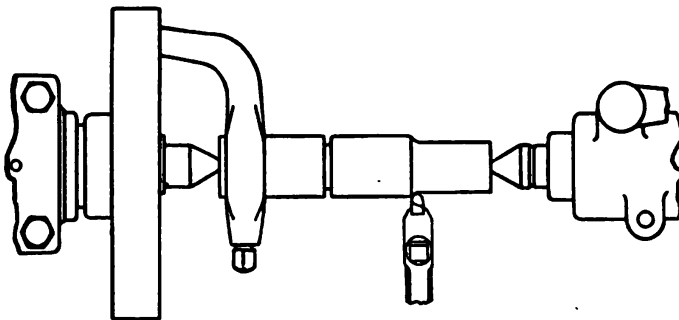


FIGURE 46.—Location of shoulder marked with parting tool.

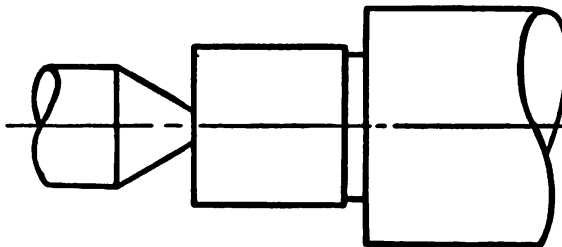


FIGURE 47.—Undercut or recessed shoulder.

k. Cutting off.—The parting tool (fig. 24) is generally used for cutting-off operations on the lathe. If the work is held between centers, do not attempt to cut through completely with the parting tool. When

only a small amount of stock remains at the center, the work will bend and squeeze the tool, damaging the work, and probably breaking the tool. Leave a sufficient amount of stock at the center and finish with a hacksaw or chisel, or remove the work from the lathe and break it off. If the work is held in some other manner, for example, in a chuck, the parting tool can be used to cut completely through the work.

17. Chucked work.—*a. Types of chucks.*—Work that cannot for some reason be set up between the lathe centers is usually held for turning in a chuck. Work that is comparatively short can be machined

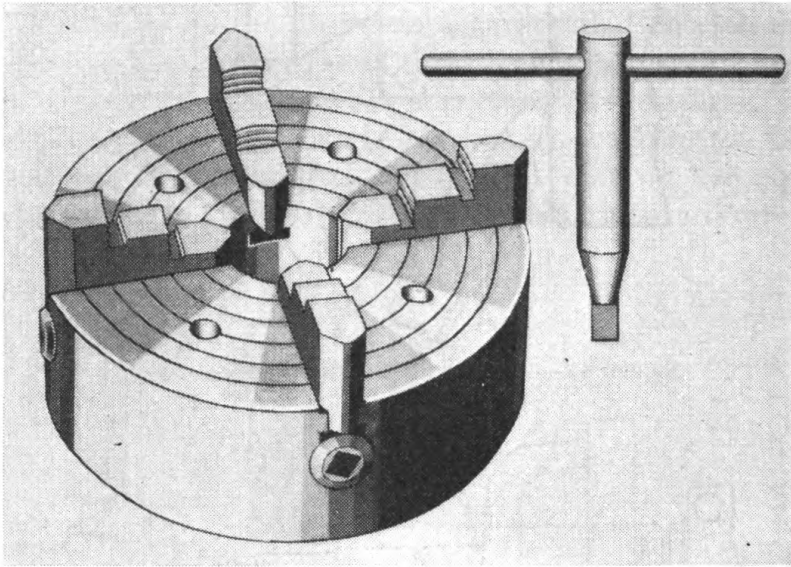


FIGURE 48.—Four-jaw independent chuck and key.

in a chuck as well as if held between centers. The following types of chucks are in general shop use:

(1) The four-jaw independent chuck (fig. 48) has four reversible jaws each of which can be adjusted independently of the others. It will hold square, round, and irregular shapes in either a concentric or eccentric position and can be adjusted to within very small tolerances. Because of its versatility and capacity for exact adjustment, it is the most commonly used type of chuck for lathe work which must be held with extreme accuracy.

(2) The three-jaw universal scroll chuck (fig. 49) has three jaws which move simultaneously and automatically center round or hexagonal work. Two sets of jaws are required for this chuck, one set for internal and one set for external chucking. Normally, it will automatically center work to within 0.003 inch of complete accuracy, but it is not general practice to use it on work where extremely accurate centering is required.

(3) The hollow headstock spindle chuck (fig. 50) is similar to a drill chuck (fig. 51) except that it is hollow and is provided with threads to screw onto the headstock spindle nose. This type of chuck is convenient for holding bars, rods, or tubes which are passed through the headstock spindle, as well as shorter pieces of small

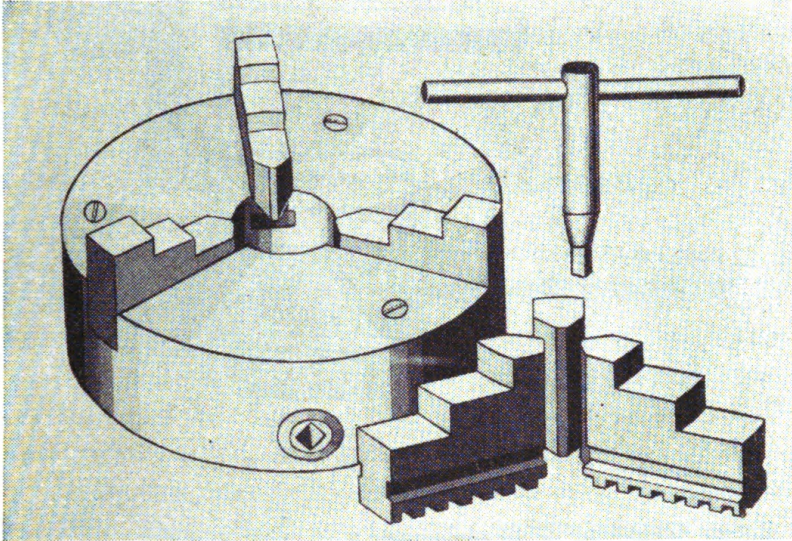


FIGURE 49.—Three-jaw universal scroll chuck and key.

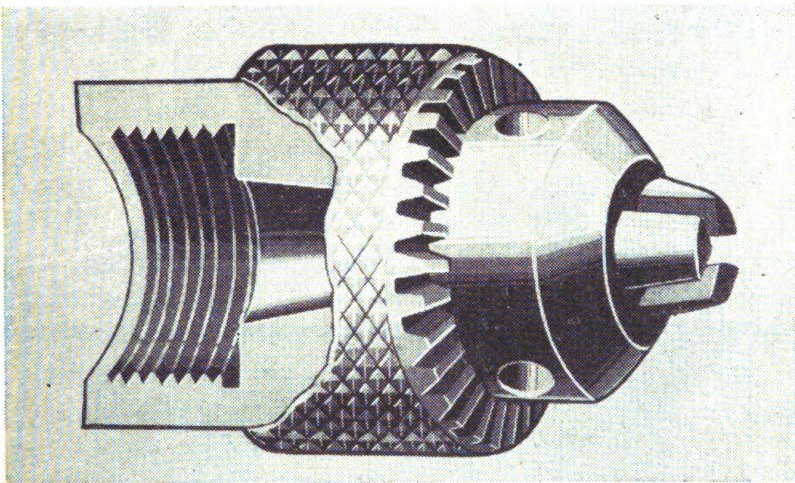


FIGURE 50.—Hollow headstock-spindle chuck.

diameter. It is more accurate than the average universal scroll chuck, being generally capable of centering work to within 0.002 inch.

(4) Drill chucks (fig. 51) can be mounted either in the headstock or tailstock of the lathe for holding straight-shank drills, reamers, taps, etc. They usually center tools within 0.002 or 0.003 inch when firmly tightened by means of the key.

b. Collets.—The draw-in collet chuck (fig. 52) is the most accurate means of holding precision work. Collets are available in a range of sizes to hold various shapes of work such as round, square, etc. Work

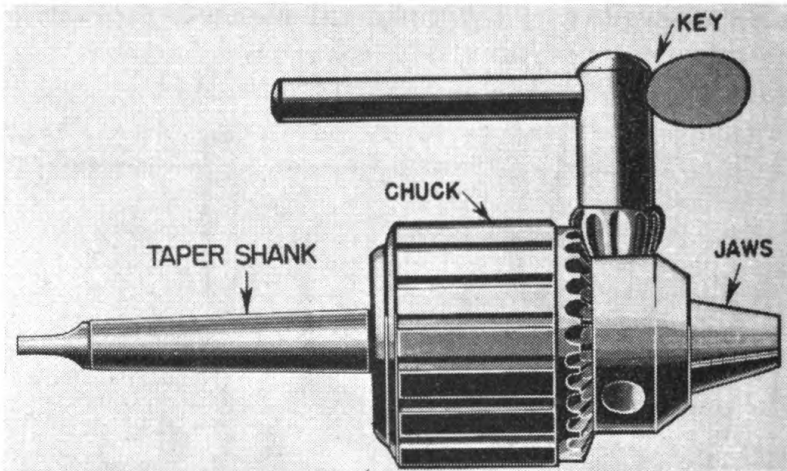


FIGURE 51.—Drill chuck and key.

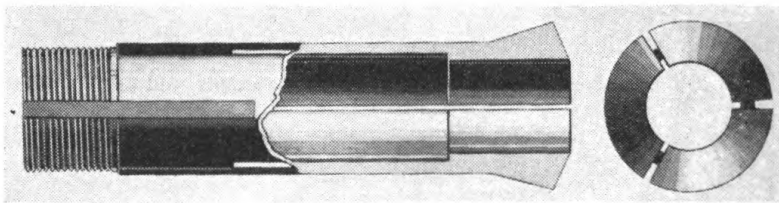


FIGURE 52.—Collet chuck.

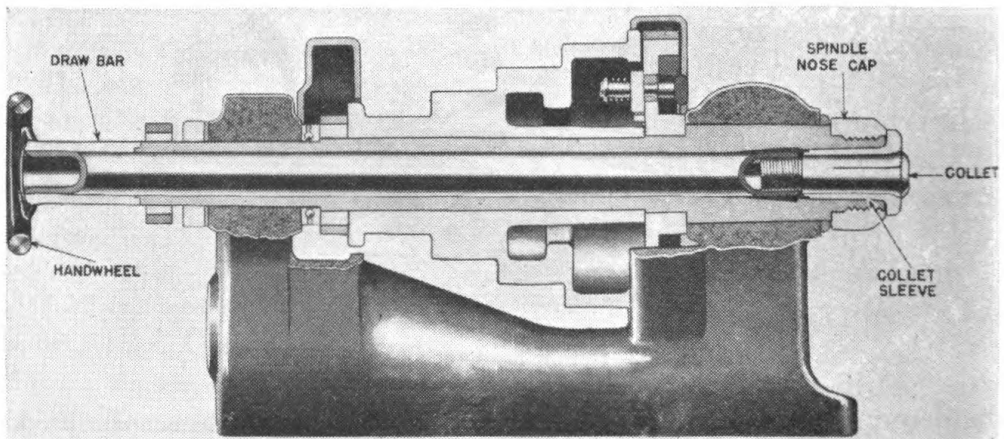


FIGURE 53.—Method of using collet chucks.

held in them should not be more than 0.001 inch larger or smaller than the specified size of the collet. Figure 53 shows the usual method of using collet chucks in the lathe. The drawbar is inserted through

the headstock spindle of the lathe and screws onto the threaded part of the collet sleeve; when the operator turns the handwheel to the right (clockwise), the collet is drawn back into the sleeve and automatically tightened on the work. Sometimes the sleeve is equipped with a lever instead of the handwheel, which enables the machinist to grip or release the work quickly without stopping the lathe, and thus save much time in the rapid production of duplicate parts.

c. Centering.—(1) Work held in the four-jaw independent chuck must be centered by adjusting the jaws and testing the results of such adjustment before a cut can be taken. The first requirement for chucking work is to select the right sized chuck; if too small a one is used, it decreases the capacity of the lathe, and if too large a one is selected, the chuck jaws may strike and damage the lathe bed, and the chuck will be awkward to use and difficult to handle. Table XIII lists practical sizes of chucks for various sized lathes.

(2) Before mounting the chuck on the lathe spindle, see that both the threads of the spindle nose and those of the chuck are thoroughly clean and lightly oiled, and that the spindle shoulder or chuckplate against which the chuck fits is free from any chips or burs. The best procedure for screwing the chuck to the spindle nose is to hold the chuck stationary and turn the lathe spindle slowly with the left hand until the chuck is screwed onto it just tightly enough to be secure. It is bad practice to run the lathe under power while mounting the chuck or to spin the chuck up to the shoulder, because it is difficult to remove the chuck.

(3) Concentric rings scribed on the faces of independent chucks allow round work to be approximately centered when it is placed in the chuck. The usual method of centering work more accurately is to start the lathe and hold a piece of chalk (preferably on a rest) against the revolving work. The chalk will mark the work clearly on the side which should be moved toward the center, and the lathe should then be stopped and the chuck jaw opposite the mark loosened slightly, and the jaw nearest it tightened a little. Repeat the test and adjustments until the chalk marks the work evenly all the way around. The work will then be centered with considerable accuracy.

(4) To center work having a smooth surface, the best method is to use a dial test indicator (fig. 7). Figure 54 shows one method of centering a hollow piece by means of a dial test indicator mounted in the lathe post. Place the contact point of the indicator against the inside diameter of the work and turn the dial until the hand points to zero. Then revolve the work slowly by hand, and the pointer on the dial will indicate not only which way the work is off center

but also how much in thousandths of an inch. To test the face of the work for "wobble" proceed as shown in figure 55, placing the contact point of the indicator against the face of the work. No satisfactory centering adjustments are possible with universal scroll chucks, hollow spindle chucks, or drill chucks; for that reason, the four-jaw independent chuck is the type most commonly used for the general run of lathe work.

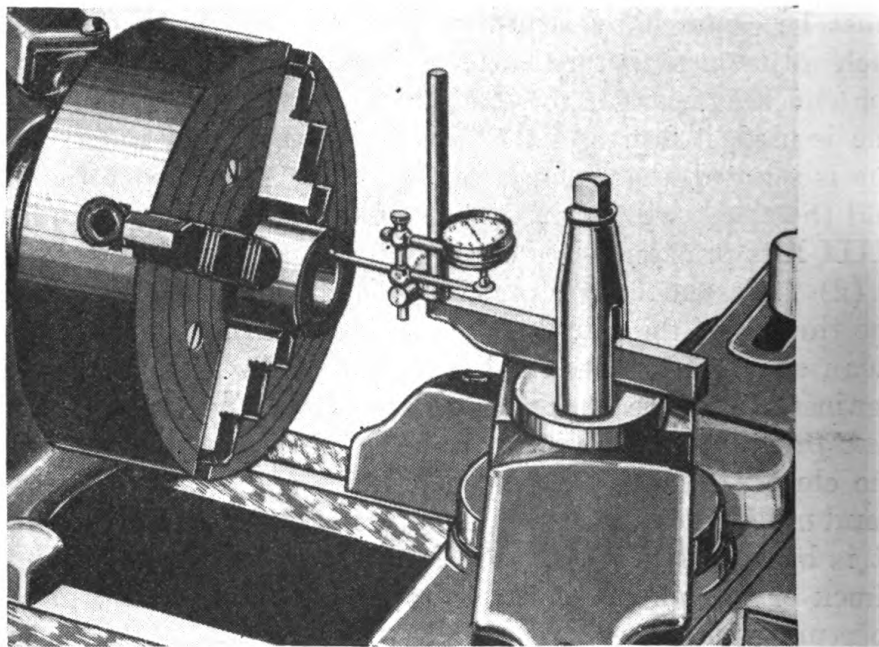


FIGURE 54.—Centering chucked work with a dial test indicator.

18. Turning and boring tapers.—*a. Characteristics of tapers.*—

(1) Taper turning is the process of machining a piece of work to a diameter which increases or decreases uniformly, thus making the piece a section of a cone. Tapers can be either external or internal; for example, a bar with its outside diameter tapered has external taper; if the walls of a hole are tapered it is an internal taper. The usual method of expressing the degree of taper of a piece of work is to specify the amount in inches that its diameter changes per foot of its length. For example, if a piece of metal 1 foot long is 3 inches in diameter at one end and 1 inch in diameter at the other its degree of taper is 2 inches per foot. To determine the taper per foot (tpf) of a given piece of work, divide the difference between the diameters at both ends in inches by the length of the piece in feet. Thus, to find the tpf of a bar 3 feet long which is 2 inches in diameter at one end and 4 inches in diameter at the other end, proceed as follows:

4 inches - 2 inches = 2 inches (difference between diameters at large and small end).

2 inches $\div$ 3 feet (length of the piece) = $\frac{2}{3}$ inch per foot.

Therefore, the tpf of the example is $\frac{2}{3}$ inch.

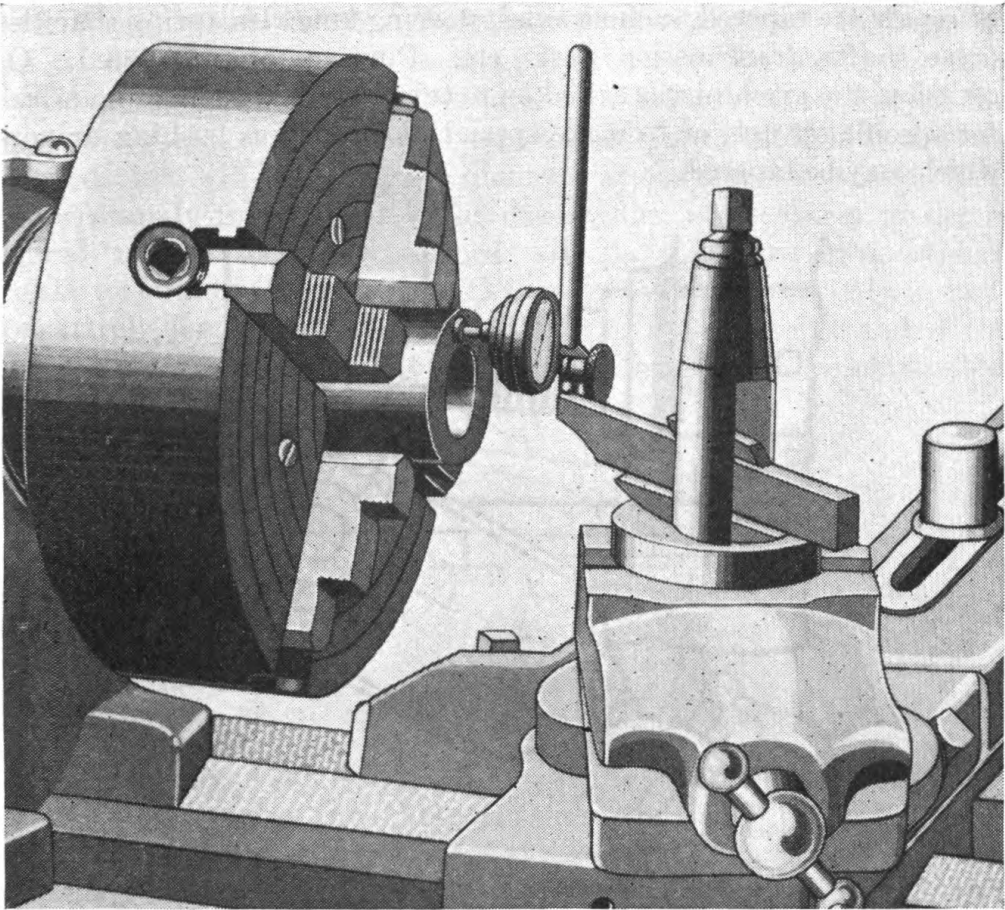


FIGURE 55.—Testing face of chucked work for “wobble” with dial test indicator.

(2) Another method of expressing the degree of a taper is to specify the included angle between the sides of the piece (*not* the angle between a side and the center line). Lathe centers taper to a point having an included angle of 60° . When turning tapers by means of the compound rest of the lathe, the operator must know the taper he wants in degrees rather than in inches per foot. The process of changing taper in degrees to tpf, or vice versa, involves rather complicated mathematical calculations which are outside the scope of this manual.

(3) As a rule, working drawings specify the taper of a piece in inches per foot if the taper is long, and in degrees if it is short

enough to be turned efficiently by means of the compound rest. In general machine-shop work, turning tapers is a common operation and worth careful study. Drill-press, lathe, and milling-machine spindles are taper bored to hold tapered tools and attachments of various kinds; a number of motor vehicle parts subject to replacement or repair are tapered, such as axles, steering knuckles, spring shackles, drive shafts, transmission parts, etc. Pins are often tapered. On occasion, the machinist is called on to turn tapered arbors or mandrels for specific set-ups, or to make special tools, such as bushing drivers, which may be tapered.

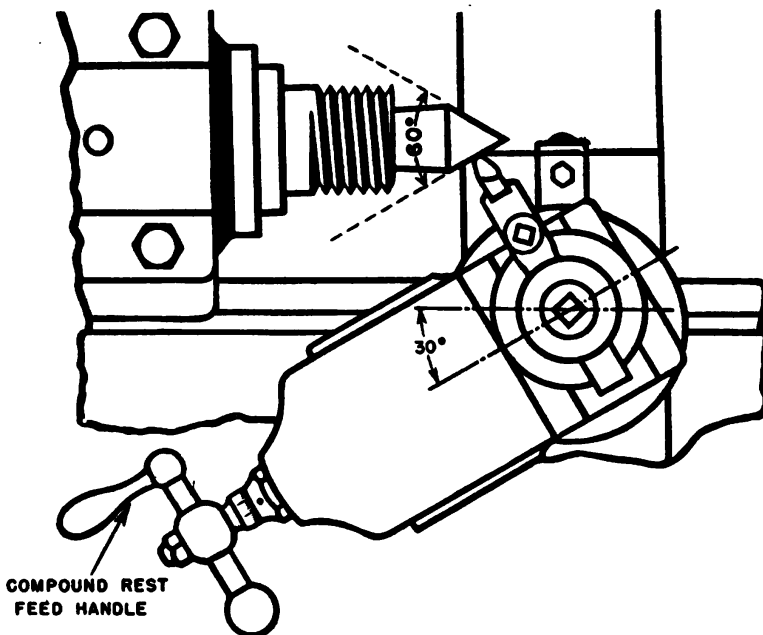


FIGURE 56.—Machining taper with compound rest.

b. Taper turning with compound rest.—One of the three generally used methods of turning tapers in the lathe is by means of the compound rest. This method is best suited to turning or boring short tapers or bevels. A good example of such turning is machining a 60° lathe center as shown viewed from above in figure 56. To machine a taper by this method, the operator sets the compound rest swivel at half the included angle and feeds the tool by hand by turning the compound rest feed handle. The direction of motion of the cutting tool coincides with the required angle of taper as shown in figure 56. Notice carefully that the angle at which the compound rest must be set is *one-half* the angle of the required taper. To machine the 60° taper being used as an example, the compound

rest is set so that the cutting tool moves at an angle of 30° toward the axis of the work. The compound rest swivel is graduated in degrees (fig. 21) so that the setting for turning various tapers is easily and quickly obtained. Taper boring and beveling can be accomplished in the same manner.

c. Taper turning with tailstock set over.—(1) Work that can be set up between centers can be tapered externally by setting the tailstock of the lathe off center. Paragraph 16*b* explained that if the live center and dead center of the machine are out of alinement, a turned piece will have a larger diameter at one end than at the other. It necessarily follows that if the dead center is set over a predetermined amount, the work turned will have a taper proportional to the amount of the set-over. The method is applicable only to comparatively long tapers, but if the work is not too short and if the dead center is carefully set over very accurate results are obtainable.

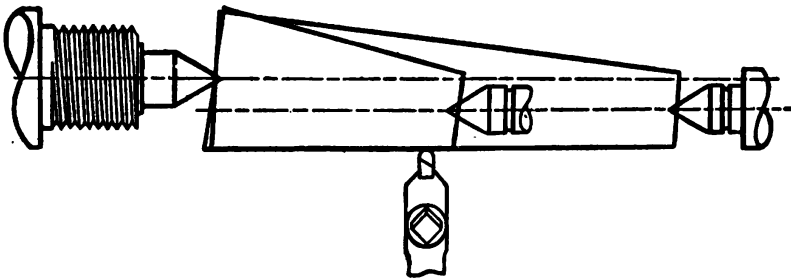


FIGURE 57.—Effect of same tailstock set-over on work of different lengths.

(2) The important thing is to set the tailstock over the right distance. The operator should become thoroughly familiar with the calculations necessary to determine that distance. Two factors affect the amount of tailstock set-over: the taper per foot desired, and the length of the work. If the set-over remains constant, pieces of different lengths will be machined with different tapers as figure 57 shows. The following are the calculations for finding the correct set-over for a given taper:

(a) Given the taper desired in inches per foot, divide the length of the work in inches by 12 (which will give its length in feet) and multiply this quotient by one-half the amount of taper per foot given (in inches). For example, to find the tailstock set-over required to machine a bar 42 inches long with a taper of $\frac{1}{2}$ inch per foot, the calculation is as follows:

$$\begin{aligned} 42 \text{ inches} \div 12 \text{ inches per foot} &= 3\frac{1}{2} \text{ feet} \\ 3\frac{1}{2} \text{ feet} \times \frac{1}{4} \text{ inch per foot} &= \frac{7}{8} \text{ inch} \end{aligned}$$

To machine the piece being used as an example with a taper of $\frac{1}{2}$ inch per foot, the tailstock should be set forward $\frac{7}{8}$ inch. If the bar were 48 inches long, the set-over would be 1 inch, as follows:

$$48 \div 12 = 4$$

$$4 \times \frac{1}{4} = 1$$

(b) To find the correct tailstock set-over, if the diameters at each end of the piece are given, divide the total length of the stock by the length of the portion to be tapered, and multiply this quotient by one-half the difference in diameters (or the difference in radii). For example, to find the tailstock set-over required to taper a bar 36 inches long with a diameter of $1\frac{1}{2}$ inches at one end, and $1\frac{3}{4}$ inches at the other, the calculation is as follows:

$$36 \text{ inches} \div 36 \text{ inches} = 1$$

$$1 \times \frac{1}{8} = \frac{1}{8}$$

To machine the piece being used as an example, therefore, the tailstock should be moved forward $\frac{1}{8}$ inch. If the piece were to be tapered only for half of its length, the calculation would be as follows:

$$36 \text{ inches} \div 18 \text{ inches} = 2$$

$$2 \times \frac{1}{8} = \frac{1}{4}$$

For this operation, the tailstock should be moved forward $\frac{1}{4}$ inch. A machinist's steel rule is the usual tool used for measuring the set-over. Figure 58 shows the procedure and is self-explanatory.

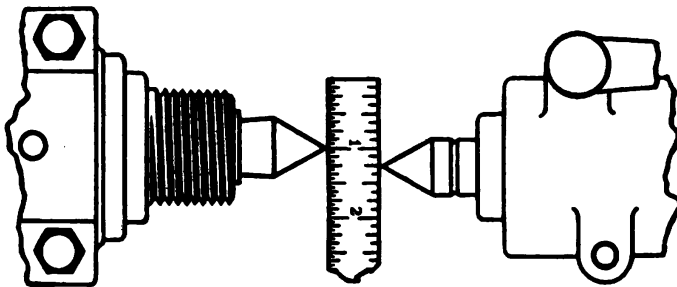


FIGURE 58.—Measuring tailstock set-over.

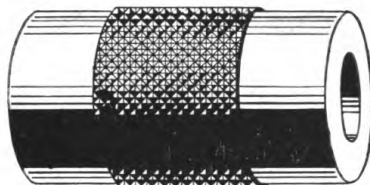
(3) Table XIV gives simple rules for figuring tapers when various dimensions are given. In all taper turning where the work is held between centers, the distance that the lathe centers enter the work must be subtracted from the length of the piece being turned. For instance, if the lathe centers enter a bar $\frac{1}{8}$ inch at each end, the length of work used in figuring the set-over must be its actual length less $\frac{1}{4}$ inch. In all taper turning operations, regardless of how the

work is set up, the cutting tool must be set on center as shown in figure 32. When turning tapers by the tailstock set-over method, the usual practice is to set the dead center forward, toward the front of the lathe, so that the small end of the work will be at the tailstock end of the lathe, and to feed the tool from right to left toward the headstock (live center).

(4) Although the tailstock set-over method is perhaps the most common way to turn external tapers, absolute accuracy is impossible because the lathe centers do not have full bearing on the work. It is necessary, therefore, that tapers so turned be tested for accuracy and any changes in the tailstock adjustment made accordingly. A satisfactory procedure for making such tests is to take a roughing cut clear across the work and test the taper either by a careful comparison of its diameters at each end or to try it in a tapered hole known to have the taper wanted. Gages similar to the ones shown in figure 59 are available for testing both internal and exter-



① Internal.



② External.

FIGURE 59.—Gages for testing tapers.

nal standard tapers. If no gage is available, the work should be tested in the hole it is to fit. To test an external taper, the best practice is to mark the piece being tested with chalk or, preferably, prussian blue and insert it snugly in the gage and turn it through one whole revolution. If the marks on the work have been rubbed equally their entire length the taper is correct; if they are rubbed at only one end of the piece the fit is inaccurate and the tailstock should be suitably adjusted before taking another cut. To test an internal taper, proceed in the same manner, except mark the gage instead of the work.

(5) Taper turning by setting over the tailstock is very convenient and satisfactory for the average run of work which does not have

to be machined within extremely close tolerances, and for tapers not exceeding 3 to 4 inches per foot. It should be borne in mind, however, that the shorter the work, the greater the inaccuracy of the taper when this method is used.

d. Taper turning with taper attachments.—(1) The third method in general use for turning tapers in the lathe is by means of either a plain or telescopic taper attachment. The method is practical for either external or internal tapers and on work set up either

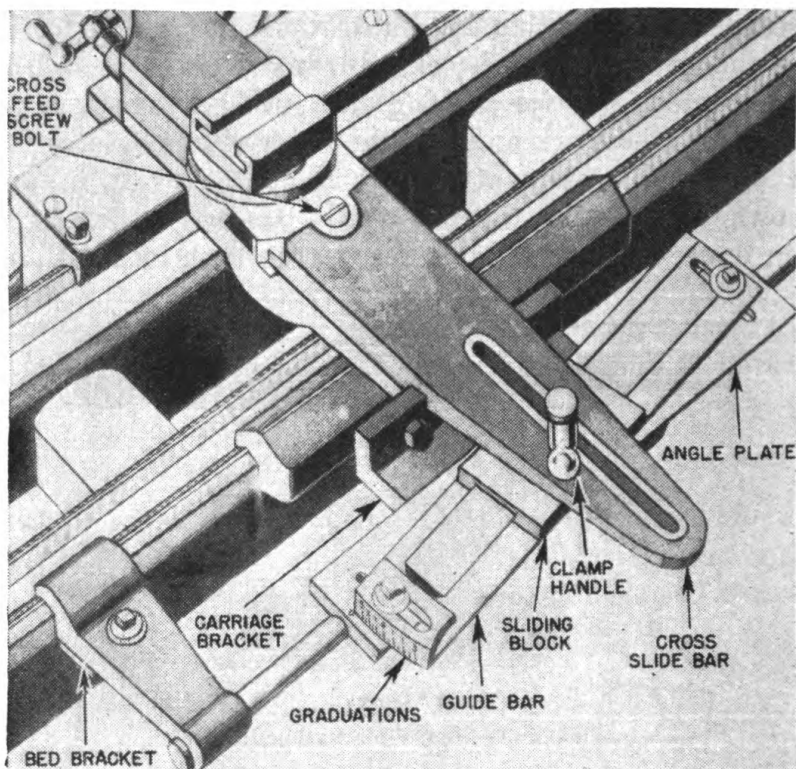


FIGURE 60.—Plain taper attachment.

between centers or in a chuck on a faceplate. Most taper attachments permit turning tapers as short as about 3 inches per foot. Figure 60 shows a popular type of plain taper attachment in place on a lathe. The angle plate is held from moving endwise by clamping the bed bracket at any desired location. The carriage bracket moves with the carriage along a dovetail on the bottom of the angle plate parallel to the ways. The sliding block moves freely along the bar. If the cross slide bar is tightened on the sliding block by the clamp handle shown, it will make the compound rest of the lathe and, consequently, the cutting tool follow the direction of the sliding block on the stationary guide bar. Before using the taper attach-

ment the cross feed screw must be disconnected by removing the cross feed screw bolt. If the guide bar is set parallel to the lathe ways, the resulting work will be straight turned; but if the guide bar is set in any direction except parallel to the ways, the work will be turned taper. It therefore follows that by setting the guide bar at a predetermined angle to the ways, a piece of work turned will have a corresponding taper. Taper attachments are graduated as shown in figure 61 so that calculations based on the length of the work are unnecessary, provided the taper desired in inches per foot is known.

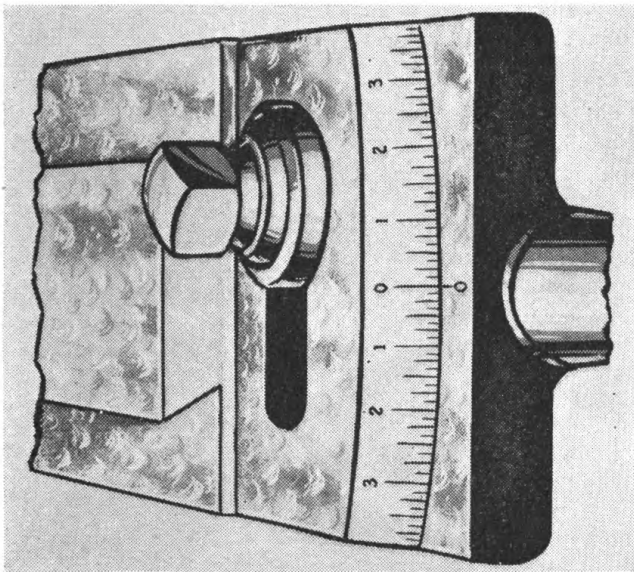


FIGURE 61.—Graduations of taper attachment.

(2) For taper turning work held between centers, the taper attachment gives very accurate results, because the lathe centers have a full bearing on the work throughout the operation. If accurate turning is called for, and a taper attachment is available, it should be used in preference to setting over the tailstock. The telescopic taper attachment is similar to the plain attachment, except that it is equipped with a telescopic cross feed screw which eliminates the necessity of disconnecting the cross feed screw. To use the telescopic attachment, first set the cutting tool for the required diameter of the work and engage the attachment by tightening the binding screws. To change back to plain turning it is necessary only to loosen these screws.

19. Drilling, tapping, and reaming.—*a. Using lathe as drill press.*—(1) The lathe can perform many drilling operations with

speed and accuracy. The most usual set-up for such drilling is to hold the work in a chuck or clamp it to the faceplate and secure a straight shank drill in a drill chuck having a taper shank that replaces the dead center in the tailstock as shown in figure 62. With such a set-up, the work is rotated by the headstock spindle and the drill fed into it by means of the tailstock handwheel. The tailstock

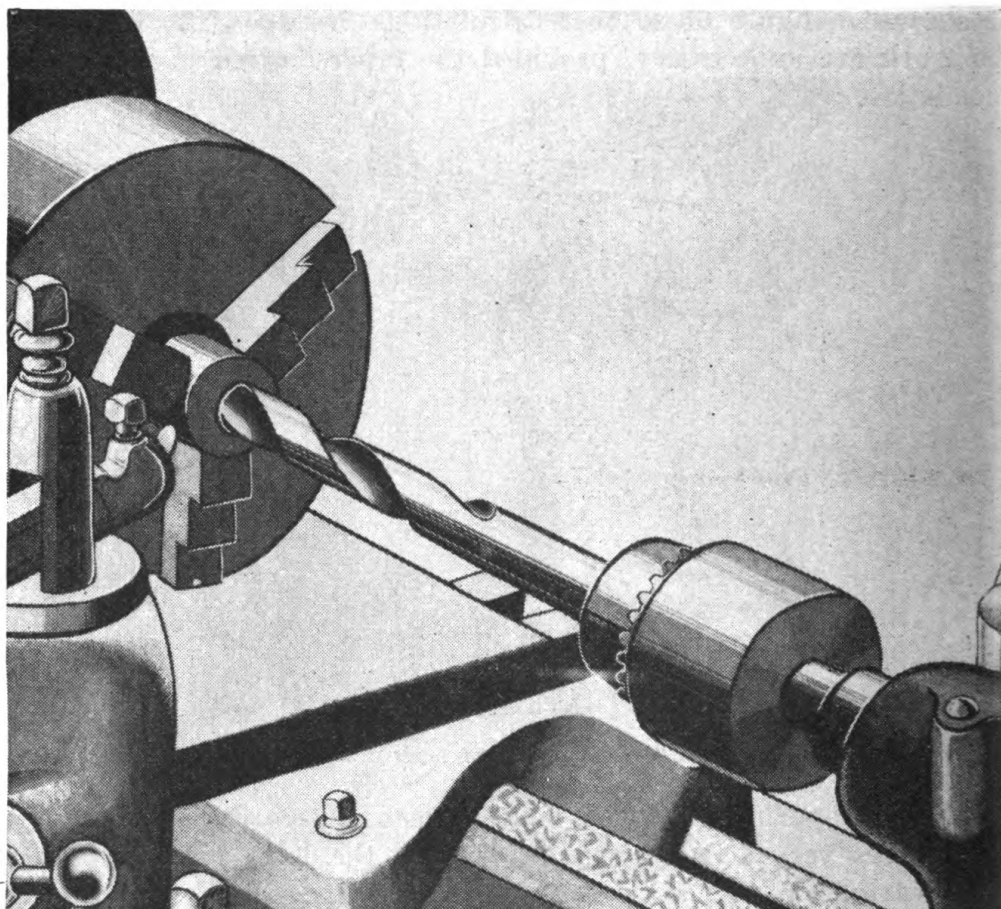


FIGURE 62.—Drilling work held in lathe chuck.

is graduated in inches and fractions of an inch so the depth of a hole can be regulated when necessary.

(2) The usual procedure in drilling by the method illustrated in figure 62 is to locate the center of the hole by methods previously described and then center punch it. The center punch mark must be accurately centered in the lathe so that the hole will be drilled at right angles to the surface of the work. The most satisfactory method of doing this centering is by means of a center indicator as shown in figure 63. Attach the indicator to the tool post of the lathe and

place its short end in the center punch mark in the work. Then move the tailstock center close to the long end of the indicator to serve as a guide and revolve the headstock spindle slowly by hand. The long end of the indicator will magnify any eccentricity of the center punch mark to a degree proportional to the difference in lengths of the ends of the indicator. If the long end of the indicator remains stationary,

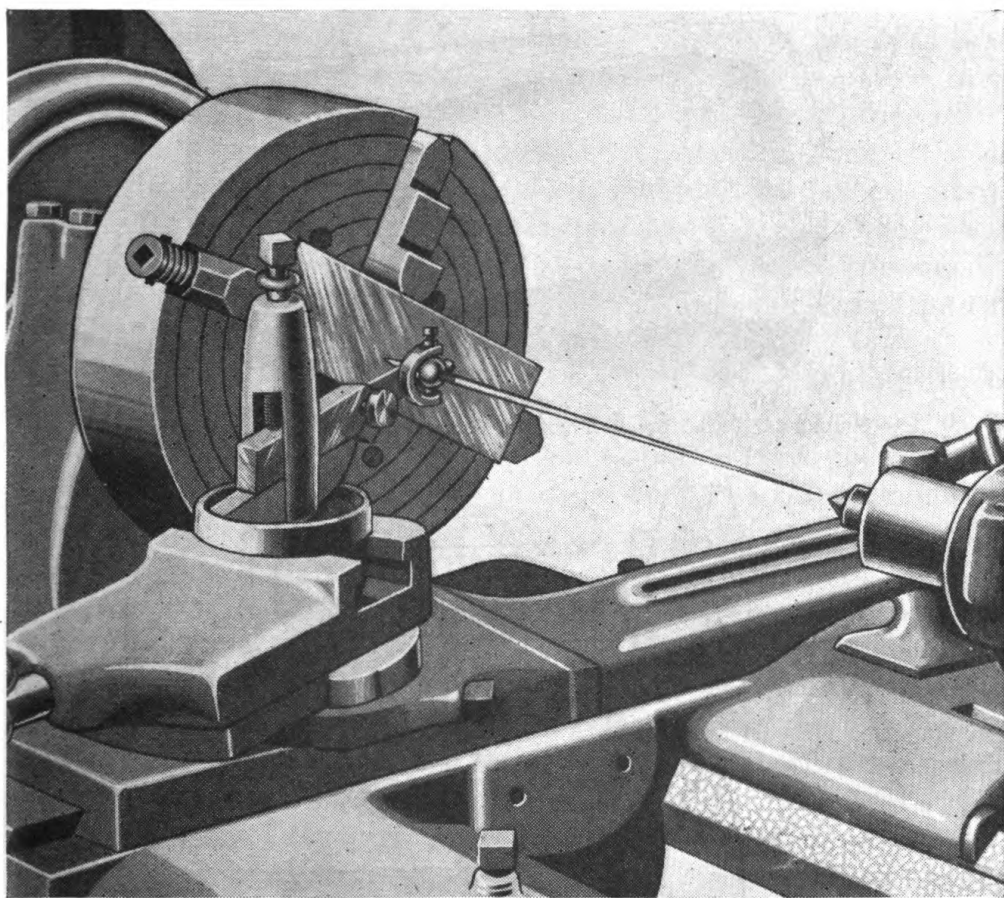


FIGURE 63.—Using a center indicator.

the center punch mark is properly centered; if the long end is not stationary, the mark is not centered and should be moved in a direction opposite to that in which the long end is eccentric. This method of centering gives very accurate results, regardless of where the hole is to be drilled in the work.

(3) If no center indicator is available, the drill can be started approximately true by clamping a tool holder in the tool post so that its butt end just touches the side of the drill as it enters the center punch mark. Sometimes, when special accuracy is desirable, center

drill the hole with a combination drill and countersink and then follow this operation with the final diameter drill.

(4) Castings with cored holes are generally drilled with a four-lip shell drill (fig. 64). Such holes should be beveled or counterbored a short distance before the actual drilling is attempted, or the drill will tend to follow the cored hole and thus be thrown off center.

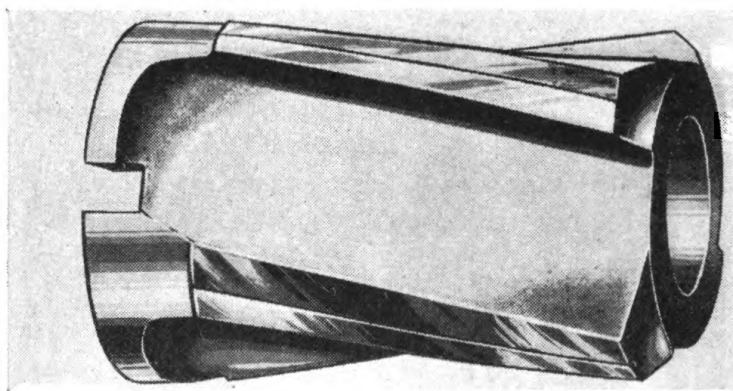
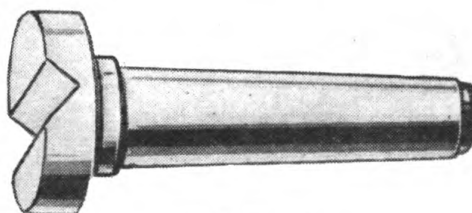
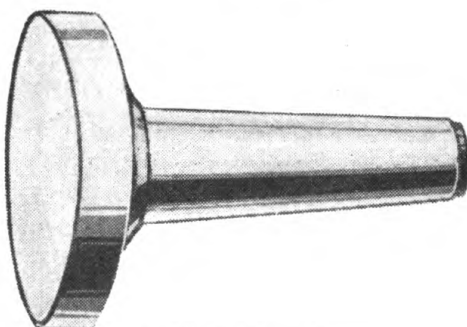


FIGURE 64.— Four-lip shell drill for drilling cored holes in castings.



① Drill pad.



② Crotch center.

FIGURE 65.—Attachments for lathe tailstock.

(5) Another method of drilling in the lathe, suitable for operations not requiring special accuracy, is to rotate the drill in a chuck mounted on the headstock spindle and hold the work against a drill pad or crotch center which fits into the tailstock spindle. Figure 65 shows

these attachments made with taper shanks to fit into the tailstock. The drill pad is used as a rest for flat work. The crotch center is grooved to hold round pieces for drilling. When drilling with either the drill pad or the crotch center, hold the work in one hand, and feed the work against the drill with the tailstock handwheel.

(6) When the drill to be used is too large for the capacity of the drill chuck, the center hole in the drill shank is placed upon the dead center of the tailstock. A lathe dog is fastened on the drill shank with its tail toward the tailstock. The tool holder is placed in the tool post in reverse, the end resting lightly against the drill to steady it. The carriage feed is placed in neutral. The lathe dog rests on the carriage along the right side of the tool holder. As the drill is fed into the work by the tailstock handwheel, the dog on the drill pulls the carriage along by the tool holder. The carriage acts as a drag upon the drill and prevents it from being drawn into the work when the drill breaks through.

b. Tapping.—Tapping is possible in the lathe by holding the tap in a chuck in the tailstock and rotating the work by means of the headstock spindle. The slowest possible spindle speed must be used, and since the tap pulls itself into the work the tailstock must be loosened so it will slide freely along the ways toward the headstock as the tapping progresses.

c. Reaming.—Except for the methods of holding the tool and the work, the principles of reaming in the lathe are the same as outlined for hand reaming, paragraph 14, TM 10-590. As a rule, reamers are held in the lathe by means of a chuck mounted in the tailstock and fed into the work by means of the tailstock handwheel. Taper shank reamers can be inserted directly into the tailstock spindle, no chuck being necessary. Reaming operations demand slow spindle speed.

20. Screw cutting.—*a. Screw thread terms and forms.*—(1) *Terms.*—Cutting screw threads is among the most common of lathe operations. Before attempting such work, however, the operator should have some familiarity with the fundamental principles of threads and the types in general use in machine shops and on motor vehicles. Such familiarity will make it comparatively easy to understand the processes and calculations involved. Throughout this and succeeding paragraphs, the following terms and definitions will be used in discussing threads and thread cutting:

(a) *Screw thread.*—A helical ridge of uniform section on the external or internal surface of a cylinder or cone.

(b) *External or male thread.*—A thread on the outside of a member.

(c) *Internal or female thread.*—A thread on the inside of a member.

(d) *Major diameter*.—The largest diameter of a screw thread.

(e) *Minor diameter*.—The smallest diameter of a screw thread.

(f) *Pitch diameter*.—The diameter of an imaginary cylinder, the surface of which would pass through the threads at such points to equalize the width of the threads and the width of the spaces cut by the surface of the cylinder.

(g) *Pitch*.—The distance from a point on a screw thread to a corresponding point on the next thread, measured parallel to the axis. Pitch in inches equals 1 divided by the number of threads per inch (N).

(h) *Lead*.—The distance a screw thread advances axially in one turn. On a single-thread screw, the lead and pitch are identical; on a double-thread screw, the lead is twice the pitch; on a triple-thread screw, the lead is three times the pitch, etc.

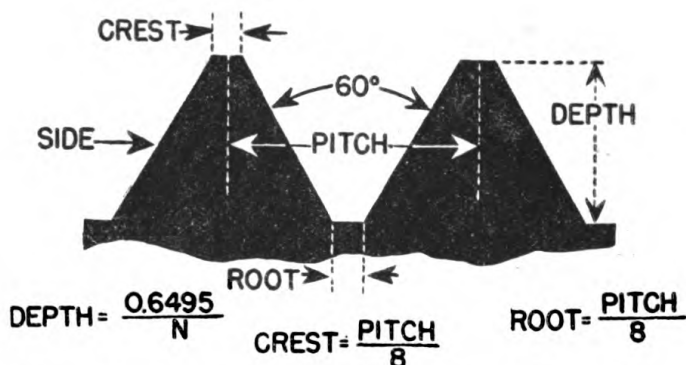


FIGURE 66.—American national screw thread forms and parts.

(i) *Thread angle*.—The angle included between the sides of adjacent threads, measured in an axial plane. It equals 60° in figure 66.

(j) *Helix angle*.—The angle formed by the helix of the thread at its pitch diameter with a plane perpendicular to the axis.

(k) *Crest, commonly called "flat"*.—The top surface joining the two sides of a thread.

(l) *Root*.—The bottom surface joining the sides of two adjacent threads.

(m) *Side*.—The surface of a thread which connects the crest and the root.

(n) *Axis of a screw*.—The center line through a screw.

(o) *Depth of thread*.—The distance between the crest and root of a thread, measured perpendicular to the axis.

NOTE.—In connection with the foregoing definitions study figure 66 carefully to become familiar with the various parts of screw threads.

(2) *Forms*.—The American national, the acme, and the square are the three most generally used forms of screw threads, and of these the American national is the most common in motor vehicle assembly (fig. 66). This form is divided for classification purposes into three series: the national coarse, national fine, and national special, usually abbreviated N. C., N. F., and N. S. The series in which a threaded part belongs depends on the number of threads it has per inch. For example, a $\frac{1}{4}$ -inch screw having 20 threads per inch is in the N. C. series; a $\frac{1}{4}$ -inch screw having 28 threads per inch is in the N. F. series; a $\frac{1}{4}$ -inch screw having 24, 27, or 32 threads per inch is in the N. S. series. Table XV lists the specifications of commonly used sizes of N. C. and N. F. screw threads, and table XVI covers the N. S. series.

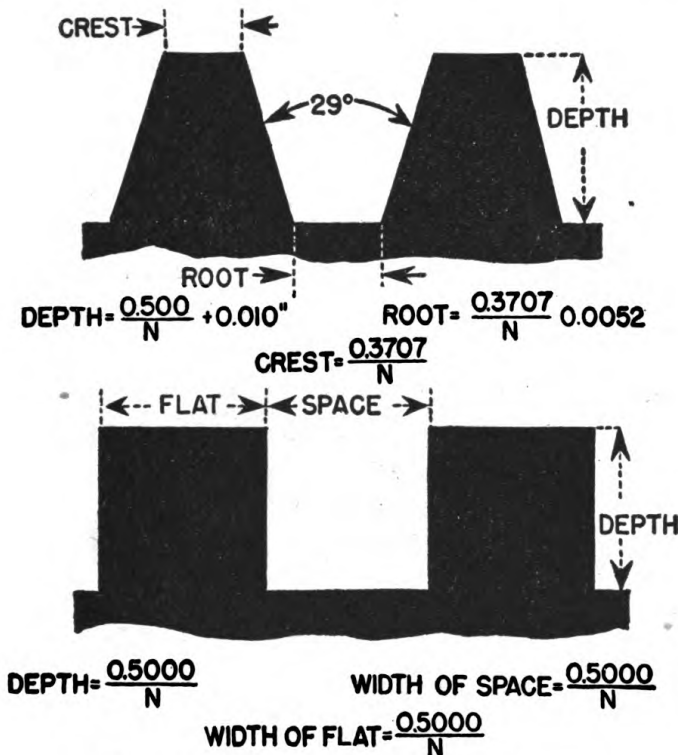


FIGURE 67.—Acme and square thread forms.

(a) The acme and square thread forms (fig. 67) are used for applications requiring exceptional strength or lasting accuracy. The acme is much more generally used than the square form. Lathe lead screws, for example, which must have strong and accurate threads are cut with acme screw threads. Vise screws, lathe cross feed screws, etc., are usually cut with square threads.

(b) The 29° worm thread (fig. 68) in either acme or Brown and Sharpe form is used for the threads in worms.

(c) The buttress thread (fig. 69) is efficient for screws which must bear an exceptional axial strain in one direction.

(d) Other types of threads less frequently used are the sharp V-thread and the Whitworth standard thread. The latter is in general use in England and some other countries.

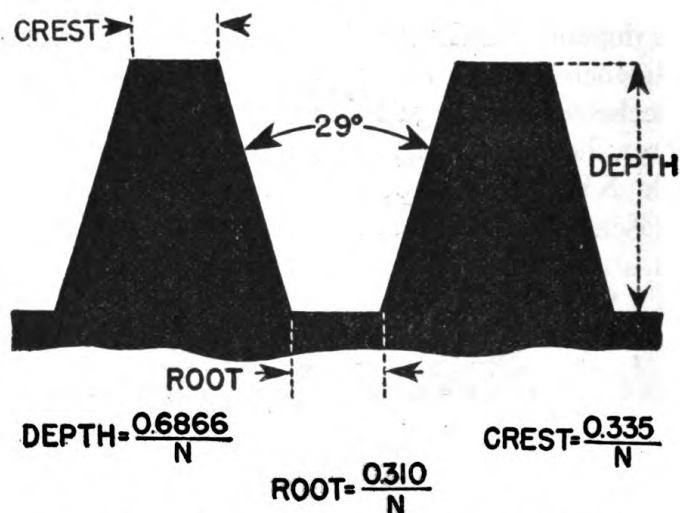


FIGURE 68.—Brown and Sharpe 29° worm thread.

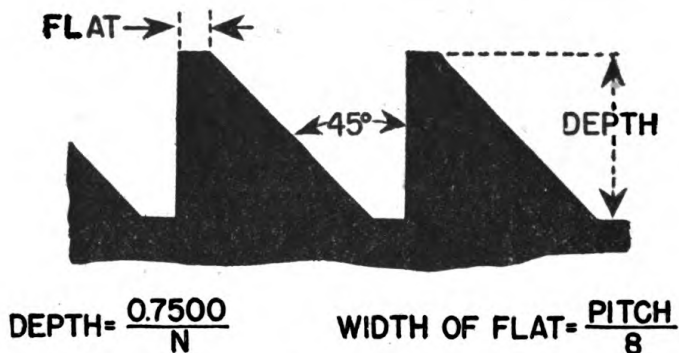


FIGURE 69.—Buttress thread form.

b. Standard change gears.—(1) Screw threads are cut in the lathe by engaging the carriage to the lead screw with the apron half nuts (figs. 22 and 23), and connecting the headstock spindle to the lead screw by a train of gears so that the carriage will travel a predetermined longitudinal distance for each revolution of the spindle. For example, if the lead screw has 8 threads per inch and the spindle revolves the work 4 times while the lead screw revolves once, the thread cut will be 4 times as fine as that of the lead screw (4×8 or 32 per inch). The gears which maintain the proportional speeds in rpm between the spindle and the lead screw are called change gears.

Figure 70 shows the standard arrangement of change gears on a screw-cutting lathe, and their usual nomenclature.

(2) The intermediate gear and the idler gear are mounted on arms in such a way that they can be moved toward or away from the lead screw gear, and they, as well as the stud and lead screw gears, can

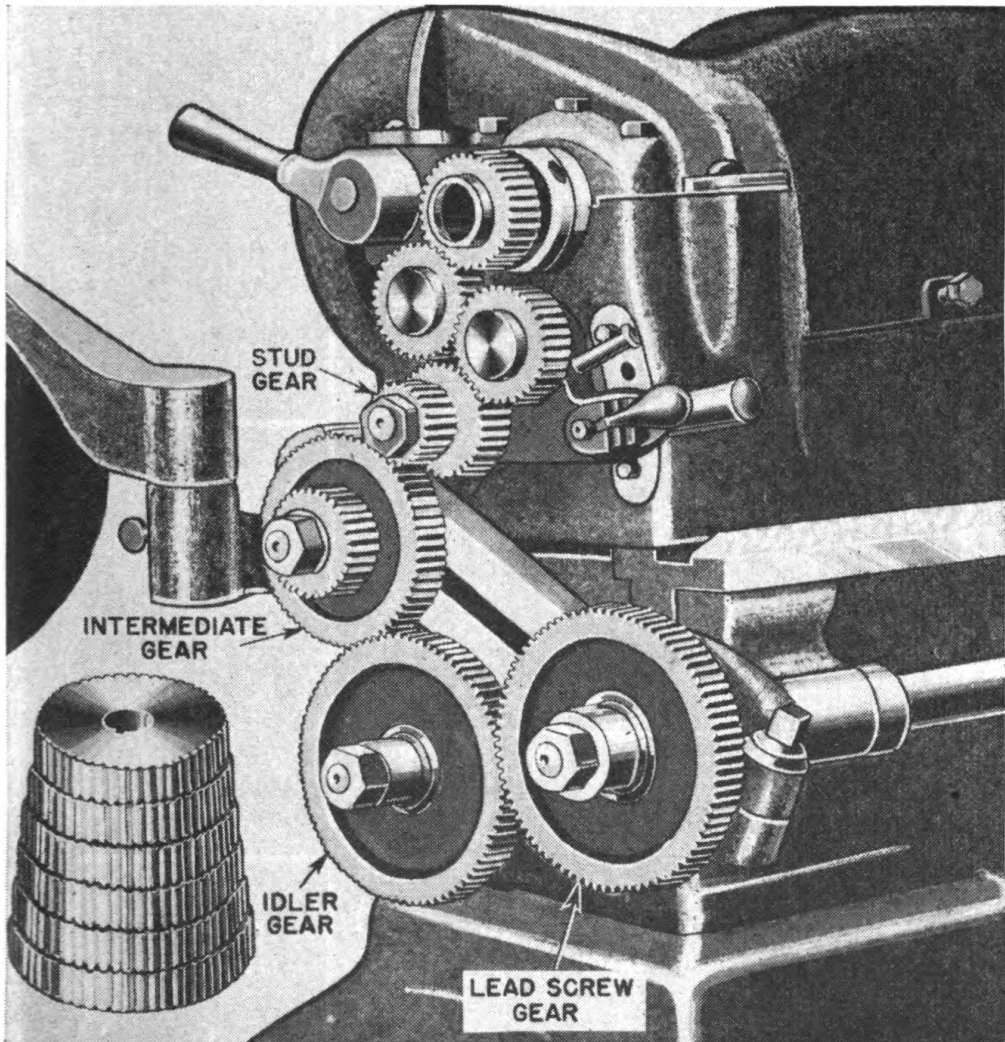


FIGURE 70.—Arrangement of standard change gears.

be removed easily and quickly from their spindles by means of a wrench. Change gear lathes are always provided with a chart similar to the one shown in figure 71, which specifies the gears the operator must use to cut a thread of some given pitch. To use such a chart intelligently requires an understanding of the fundamental principles which it involves.

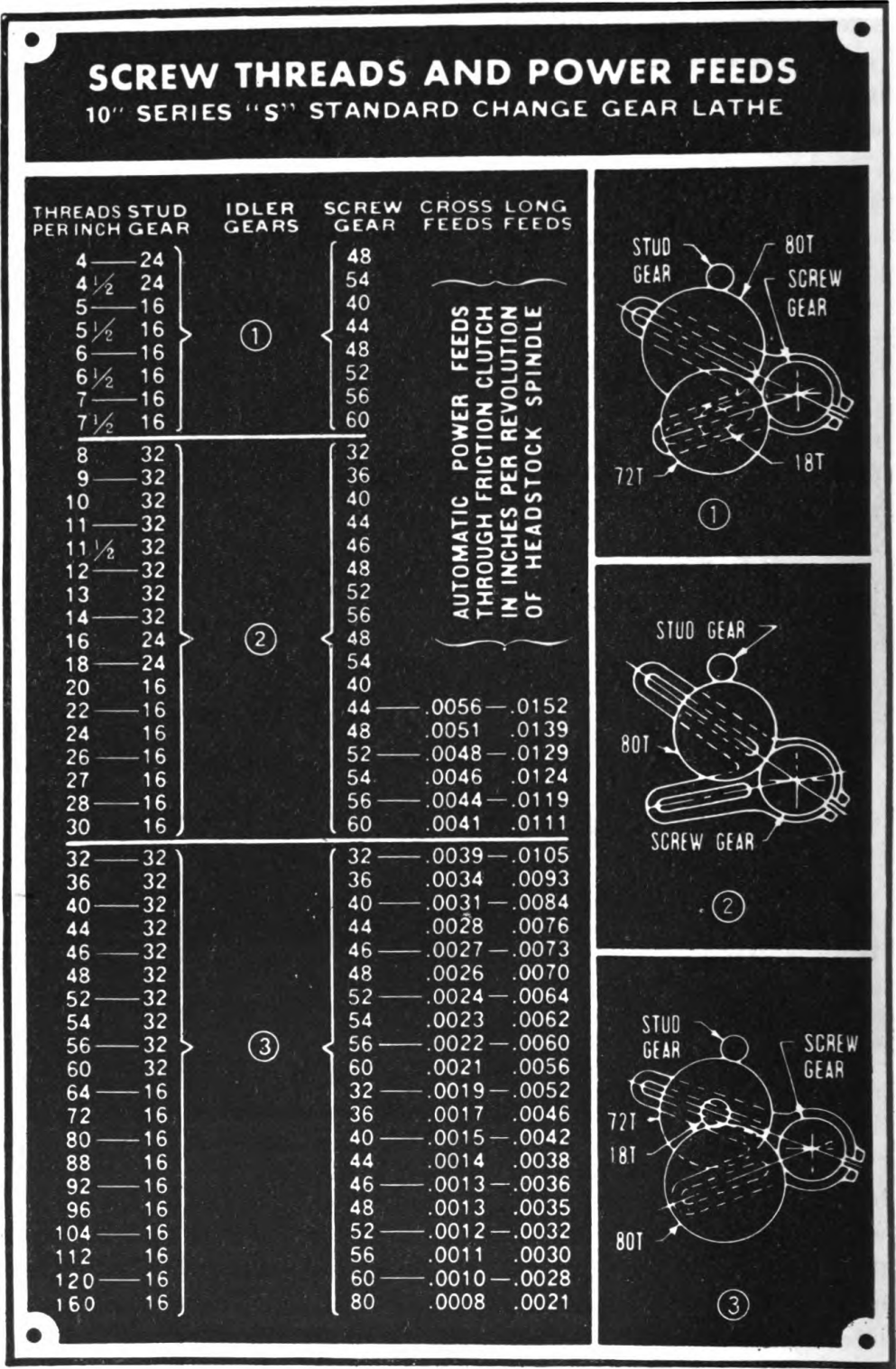


FIGURE 71.—Change-gear chart for standard change-gear lathes.

(3) The number of teeth in the stud gear and the lead screw gear, and in the intermediate gear if one is used, determines the pitch of thread that will be cut. Gear arrangements which do not make use of the intermediate gear are known as simple gearing; the stud gear is connected directly to the lead screw gear through the idler gear as shown in figure 71②. The rule for determining the number of teeth of the stud gear and lead screw gear in a simple train in order to cut a given pitch of thread is as follows: Multiply the number of threads per inch of the lead screw and the number of threads per inch to be cut by the same number. The products will be the number of teeth that the stud gear and lead screw gear should have respectively. For example, suppose that a machinist wants to cut a screw with 10 threads per inch on a lathe having a lead screw with 4 threads per inch. The procedure would be to multiply 10 and 4 by any convenient number, say 6. Then $6 \times 4 = 24$ and $6 \times 10 = 60$. The stud gear should then have 24 teeth and the lead screw gear 60 teeth. If gears with 24 and 60 teeth are not available, the machinist multiplies 4 and 10 by some other number which will give the numbers of teeth of the gears that are at hand. (Whenever the thread to be cut is finer than the thread of the lead screw, the gear with the fewest teeth is the stud gear, and the gear with the largest number of teeth is the lead screw gear).

(4) As an example of the use of the change-gear chart (fig. 71), suppose the machinist wants to cut a screw with 16 threads per inch. The procedure is first to find 16 in the left-hand column of the chart under "threads per inch." Following across to the right, the stud gear (second column) should have 24 teeth. The idler gear must be arranged as shown in figure 71②, that is, it must have 80 teeth and be so placed on the arm provided that it will engage both the stud gear and the lead-screw gear. Column 4, under "screw gear" then specifies that the lead screw must have 48 teeth. Note on the chart that all pitches of threads from 8 through 30 per inch can be cut by the simple gearing shown in figure 71②. The last two columns on the right of the chart specify the feed in fractions of an inch, both cross and longitudinal, that the gearing will provide. For example, if gears are set as specified for cutting 28 threads per inch, and the power longitudinal feed is used for a straight turning operation, the cutting tool will move 0.0119 inch longitudinally during each revolution of the spindle. The power cross feed will move the cutting tool 0.0044 inch per revolution. These feeds have nothing to do with thread cutting.

(5) It frequently happens that for various reasons it is impossible to obtain a desired number of threads per inch by simple gearing. For example, if it is desired to cut 80 threads per inch with a lead screw having 8 threads per inch, the lead screw gear must have 320 teeth, which would make it too large in diameter to fit the lathe. By compounding the gears, it is possible to cut 80 threads per inch with the gears generally available. In a compound train, an additional intermediate gear is provided as shown in figure 70. Usually it consists of two gears of different size mounted on the same spindle so that they revolve together; when so arranged, the larger gear is called the first intermediate gear, and the smaller gear is called the second intermediate gear. (Some machinists refer to these gears as the first and second idler gears.) One convenient rule for figuring compound gears is as follows: Establish the ratio between the number of threads per inch to be cut and the number of threads per inch of the lead screw, just as for simple gearing; then factor both terms of the ratio and multiply the factors by any convenient number. The result will be the number of teeth required on the stud, intermediate, and lead screw gears. For example, to cut 80 threads per inch on a lathe having a lead screw with 8 threads per inch, the calculation is: $\frac{80}{8} = \frac{2 \times 4}{8 \times 10}$ (factors). Then, multiplying 2, 8, 4 and 10 by any convenient number, say 12, the result is: $\frac{24 \times 48}{96 \times 120}$. And the gearing must be:

	<i>Teeth</i>
Stud gear-----	24
First intermediate gear-----	96
Second intermediate gear-----	48
Lead screw intermediate gear-----	120

(6) Figure 71① and ③ shows the arrangement of compound gears necessary to cut threads from 4 through $7\frac{1}{2}$ per inch and from 32 through 160 per inch. To cut 6 threads per inch, select a stud gear having 16 teeth, and a lead-screw gear having 48 teeth. Then arrange these as shown in figure 71① so that the stud gear drives the idler gear, which drives the first intermediate gear. The second intermediate gear revolves with the first and drives the lead screw gear. For cutting all threads from 4 through $7\frac{1}{2}$ per inch, the idler gear has 80 teeth, the first intermediate gear has 18 teeth, and the second intermediate gear has 72 teeth. The stud gear and the lead screw gear only need be changed, and the idler and intermediate gears moved on the arms provided so that the whole train is in

mesh. Figure 71 ③ shows the method of gearing to cut threads from 32 through 160 per inch. For example, to cut 60 threads per inch, select a stud gear having 32 teeth, and a lead screw gear having 60 teeth. Then arrange these as shown in figure 71 ③ so that the stud gear drives the first intermediate gear; the second intermediate gear revolves with it and drives the idler gear, which in turn drives the lead screw gear. For cutting all threads from 32 through 160 per inch, use the same idler and intermediate gears as for threads from 4 through $7\frac{1}{2}$ per inch but reverse their positions in the train. The change gear charts of different makes of lathes may differ in small details but the fundamental principle is the same in all of them, and you should have no difficulty in selecting and arranging change gears for any pitch of thread which the lathe in use is capable of cutting.

c. Quick change gears.—(1) Quick change gear lathes are provided with a mechanism which permits screw threads of various pitches to be cut without changing any loose gears or making any calculations. Figure 72 shows a lathe equipped with a quick change gear box and controls; figure 73 shows an enlarged view of an index plate, always a part of such a mechanism.

(2) The proper gearing to cut a thread of a desired pitch is obtained by shifting the sliding gear in or out, and by moving the top lever to the left, right, or center position as indicated by the index plate. The tumbler lever is always locked under the column in which the required number of threads occurs. For example, to cut 24 threads per inch, first find 24 on the index plate (fig. 73), then, according to the chart (fig. 71), see that the sliding gear is in the "in" position (first column) and that the top lever is moved as far as possible to the right (second column). Then lock the tumbler lever under the column of the plate in which 24 occurs, and the lathe is geared to cut 24 threads per inch (or to feed 0.0140 inch per spindle revolution longitudinally). To cut 16 threads per inch, the arrangement is the same except that the tumbler lever is locked under the column in which 16 appears. The same procedure applies to any pitch within the capacity of the machine.

d. Thread cutting tools.—The thread cutting tool illustrated in figure 24 is the one used for cutting American national form screw threads. Because its point is ground to an included angle of 60° , it automatically forms the thread correctly. However, this tool, as well as any tool used to cut threads, must be very carefully and accurately ground, or the results will not be satisfactory. The tool for cutting American national form threads usually has no top

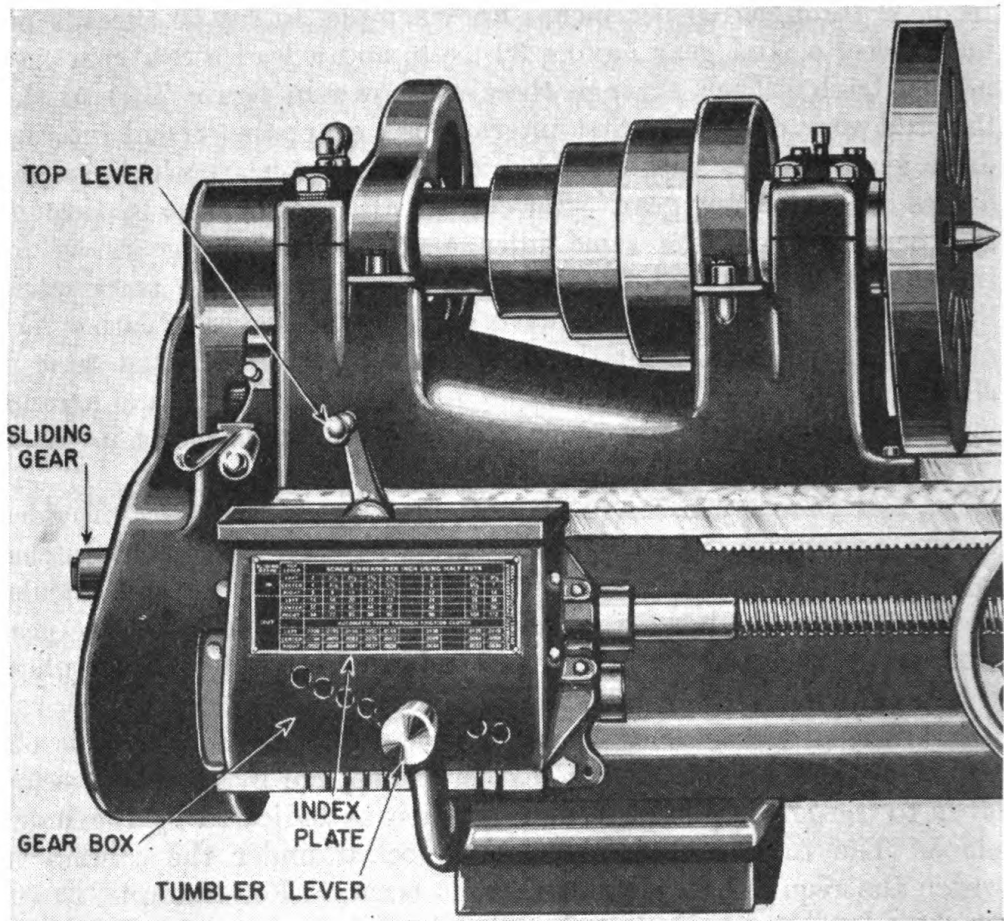


FIGURE 72.—Quick change gear mechanism.

16-INCH QUICK CHANGE GEAR LATHE										
SLIDING GEAR	TOP LEVER	THREADS PER INCH—FEEDS IN THOUSANDTHS								
IN	LEFT	4	4½	5	5½	5¾		6	6½	7
	CENTER	.0841	.0748	.0673	.0612	.0585		.0561	.0518	.0481
	RIGHT	8	9	10	11	11½		12	13	14
OUT	LEFT	.0421	.0374	.0337	.0306	.0293		.0280	.0259	.0240
	CENTER	16	18	20	22	23		24	26	28
	RIGHT	.0210	.0187	.0168	.0153	.0146		.0140	.0129	.0120
OUT	LEFT	32	36	40	44	46		48	52	56
	CENTER	.0105	.0093	.0084	.0076	.0073		.0070	.0065	.0060
	RIGHT	64	72	80	88	92		96	104	112
OUT	LEFT	.0053	.0047	.0042	.0038	.0037		.0035	.0032	.0030
	CENTER	128	144	160	176	184		192	208	224
	RIGHT	.0026	.0023	.0021	.0019	.0018		.0017	.0016	.0015
AUTOMATIC CROSS FEED EQUALS .375 TIMES LONGITUDINAL FEED										

FIGURE 73.—Quick change gear index plate.

rake and about 5° front clearance. A center gage (fig. 74) is convenient for checking the 60° tool angle. A formed threading tool (fig. 75) is sometimes used in a special holder when considerable threading is to be done. It is so formed that the machinist can sharpen it by grinding it on top only. Tools for cutting square, acme, and worm thread forms must be specially ground. *In all thread-cutting operations, the cutting tool must be set on center as shown in figure 32.*

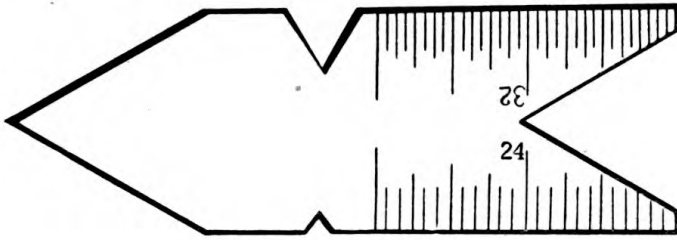


FIGURE 74.—Center gage.

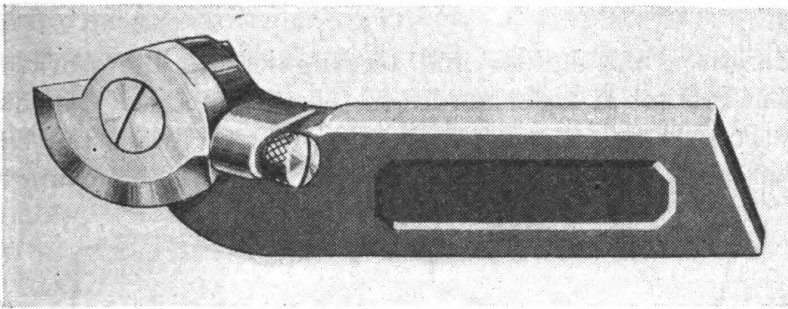


FIGURE 75.—Formed threading tool.

e. Thread cutting.—(1) Screw threads can be either right-hand or left-hand and straight or tapered. A screw with a right-hand thread advances when turned to the right, or clockwise; a left-hand screw advances when turned to the left, or counterclockwise. To cut right-hand threads, feed the cutting tool from right to left, toward the headstock; to cut left-hand threads, feed from left to right, toward the tailstock. When the lathe has been properly geared for the number of threads per inch desired, and the tool correctly ground, the next step is to make sure that the tool is set on center, and then see that its axis is set at exactly 90° to the axis of the work. This angle can be checked quickly with a center gage by putting one side of the gage along the outside of the work and then adjusting the tool point until it exactly fits the notch on the other side of the gage as shown in figure 76.

(2) The next step is to determine the setting of the compound rest. As a general rule, when cutting threads of fine pitch, 30 per inch and finer, satisfactory threads can be obtained by holding the cutter bit

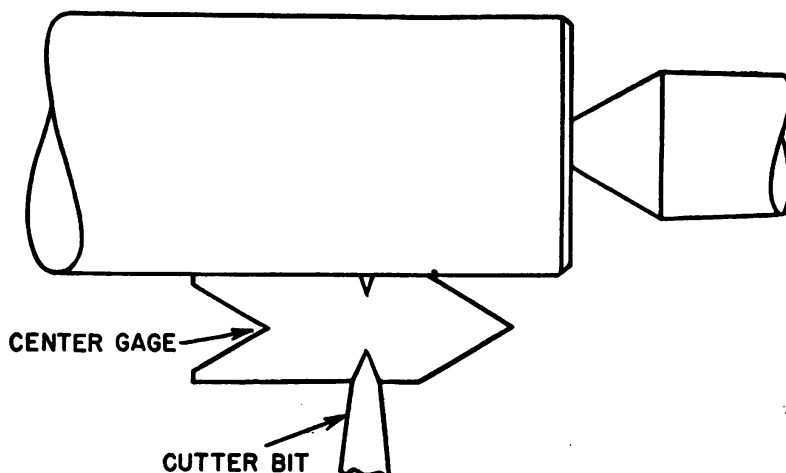


FIGURE 76.—Using center gage to set threading tool.

in a straight shank holder and feeding it into the work with the compound rest set at right angles to the work's axis. To cut threads coarser than 30 per inch, it is better practice to set the compound rest at an angle slightly less than one-half the included angle of the

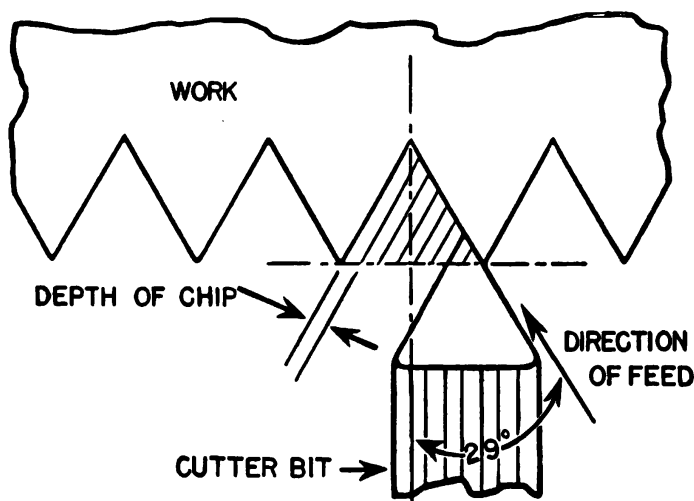


FIGURE 77.—Thread cutting with compound rest at 29° angle.

thread, with the cutter bit held in an offset tool holder but still at right angles to the axis of the work as shown in figure 77. Some machinists set the compound rest at 29°, others at 29½° or even 30°

with equal results. This method feeds the tool into the work so that it cuts only on one side of the thread. This feed is excellent for roughing cuts; for the last two or three finishing cuts, however, the tool should be fed straight in to remove any tool marks and give the thread a good finish.

(3) The usual practice in cutting threads is to take a very light initial cut and then check to see that the lathe has been geared for the right number of threads per inch. If it is correctly geared, then continue taking cuts along the thread until it has the depth wanted, usually at a depth of cut of about 0.003 to 0.005 inch for each traverse of the tool. Table XVII gives the number of cuts usually required to cut threads of commonly used pitch. At the end of each cut, the half nuts are usually disengaged and the carriage is returned to position for the next cut by hand. Some method must be provided, therefore, to engage the half nuts for the following cut at a point on the lead screw which will cause the cutting tool to follow the previous cut.

(4) The usual device for accomplishing this is a chasing dial, which is supplied with all screw cutting lathes. It consists of a worm wheel which meshes with the lead screw, and a short shaft which connects the worm wheel to the dial. The dial is calibrated with four numbered lines and four others between them, not numbered, as shown in figure 78. Its use is not complicated: when cutting an even number

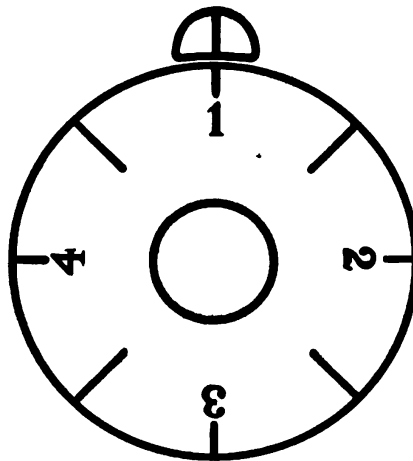


FIGURE 78.—Chasing dial.

of threads per inch, the half nuts can be re-engaged for following cuts when *any* line on the dial is opposite the index mark; when cutting odd-numbered threads, the half nuts can be closed when any *numbered* line is opposite the index mark; to cut all threads

having half a thread per inch (as for example $11\frac{1}{2}$ threads per inch) engage the half nuts when any *odd-numbered* line is opposite the index mark. Some lathes are equipped with a chasing stop which is bolted to the lathe carriage and regulates the depth of cut for each tool traverse, or can be set to limit the total depth of cut of the thread. If a lathe with a chasing stop is used, refer to the manufacturer's manual for detailed instructions as to its use. When a thread is cut, the end of it must often be finished in any one of several ways. The two most common are to finish it with a 45° chamfer or to round it with a specially ground forming tool as shown in figure 79.

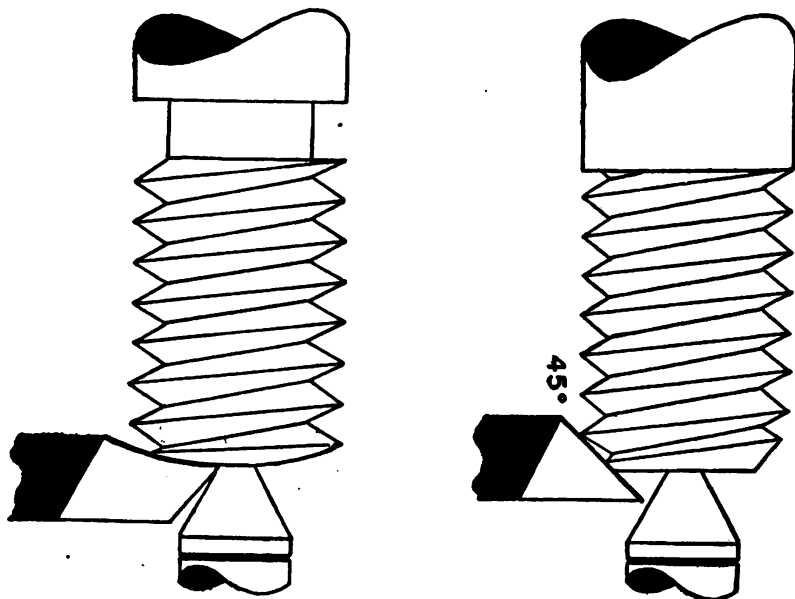


FIGURE 79.—Two methods of finishing ends of threads.

f. Taper screw threads.—The machinist can cut taper screw threads or pipe threads in the lathe either by setting over the tailstock or with a taper attachment as described in paragraph 18*d*. In cutting such threads, the cutting tool must be set at right angles to the axis of the work exactly as though the threads were straight. Do *not* set the tool at right angles to the taper of the thread.

g. Internal screw threads.—Inside threads are cut on the lathe by an inside threading tool (fig. 24) held in a boring bar. The procedure is the same as for cutting outside threads; internal threads can be either straight or taper. Paragraph 23 describes the use of boring bars in detail.

h. Measuring threads.—(1) The pitch or the diameter of a threaded part must often be measured with close accuracy. Any one

of several methods and tools can be used. A machinist's rule or a screw pitch gage is convenient for determining the pitch of any thread. Figure 80 shows the use of both to measure the number of threads per inch of a screw. When measuring threads per inch with a rule, do *not* count the thread at the end of the rule. The screw illustrated has 8 threads per inch. The screw pitch gage is simply a number of steel blades notched to fit several different pitches of

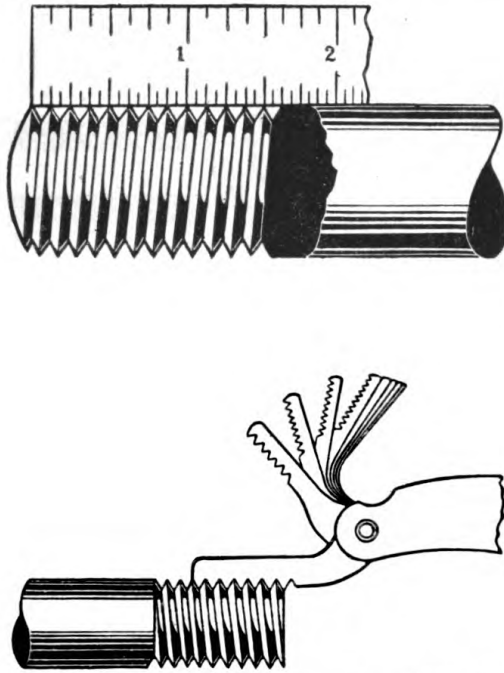


FIGURE 80.—Two methods of measuring pitch of screw.

screws. Find the one that will exactly fit the threads of a given screw and read the pitch direct from the gage.

(2) The measurement of the thread angles and pitch diameters is somewhat more involved but equally necessary if accurate fits are to be obtained. The simplest way to check the accuracy of a screw thread is to try it in the part in which it is to fit. If this is impractical, a plug thread gage or a ring thread gage (fig. 81) can be used to test the part. The usual method of measuring the pitch diameter of threads is by means of a thread micrometer, which is similar to any other micrometer except that the spindle and anvil are formed to fit the screw thread as shown in figure 82. The accuracy of the measurement depends upon the thread being cut to exact form.

21. Eccentric turning.—*a. Principles.*—(1) Eccentric means “off center.” Applied to machine-shop work, if a piece of work is machined in such a way that it has a part turned on a different center from its main part, the part off center is said to be eccentric. For example, the main journals of the crankshaft shown in figure 83 are on a center line through the ends of the shaft. The crankpin, is on a center line drawn through its own center, and so is eccentric



FIGURE 81.—Thread gages.

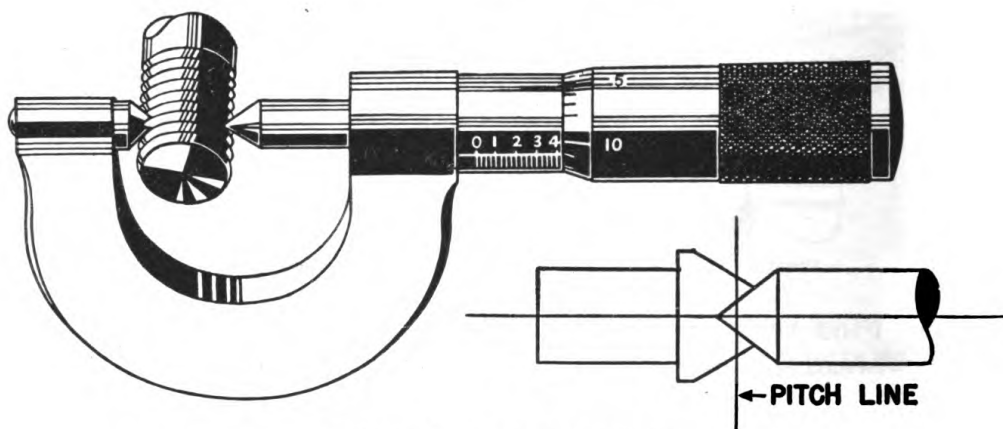


FIGURE 82.—Thread micrometer.

to the main journals. To machine the main journals of such a shaft in the lathe, countersink work centers in the ends of the shaft in the usual manner as shown in the lower part of figure 83. To machine the crankpin, however, provide work centers in line with its center as shown in the upper part of figure 83.

(2) Eccentric turning in the lathe is similar to any other kind of turning, except that work centers must be provided at each end of and in alinement with the center of each surface to be machined. For heavy work, counterweights must sometimes be provided to insure smooth and accurate operation.

b. Drilling an eccentric hole.—As a simple example of machining eccentric work in the lathe, suppose the operator wants to drill a $\frac{1}{4}$ -inch hole through a piece of steel 6 inches in diameter, and have the center of the hole 2 inches from the center of the piece. If the work is chucked in the usual manner, the hole will be in the center of the work not 2 inches off. The problem therefore is to set up the job so the point to be the center of the hole is on a line between the lathe centers, regardless of where the center of the piece is. Since such a set-up cannot be obtained conveniently by means of a chuck, the usual alternative is to center punch the point to be the center of the hole and clamp the work to a large faceplate so that the center punch mark

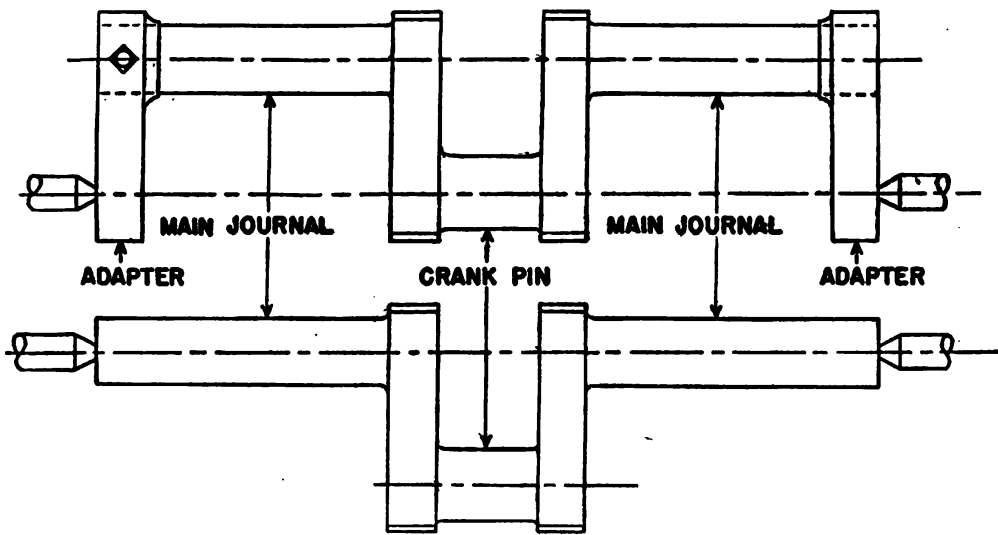


FIGURE 83.—Example of eccentric machining.

is on a line between the lathe centers. A center indicator as shown in figure 63 and explained in paragraph 19*a* is convenient for accurately centering the punch mark. When the punch mark is accurate, drill the hole in the usual manner to obtain the result shown in figure 84.

c. Machining a crankshaft.—Figure 83 illustrates and paragraph 21*a* discusses briefly the operation of turning a crankshaft. The job is a common one in automotive machine shops. As pointed out in *a* above, eccentric turning differs from straight turning only in the set-up of the work. The part of the work actually being cut must be between the lathe centers; the position of any part of it not being worked can be disregarded. As shown in the lower part of figure 83, the main journals are the part of the crankshaft actually being cut; it is immaterial where the crankpin is, so long

as the main journals are centered. When the main journals are turned, reset the work so that the crankpin is between centers, as shown in the upper part of figure 83. It then becomes immaterial where the main journals are. If the throw of the eccentric is less than the radius of the shaft, both pairs of work centers could be countersunk in the shaft end; in the example being used, the throw is more than the radius of the shaft, so work centers must be provided in line with the center of the crankpin. Some crankshafts are

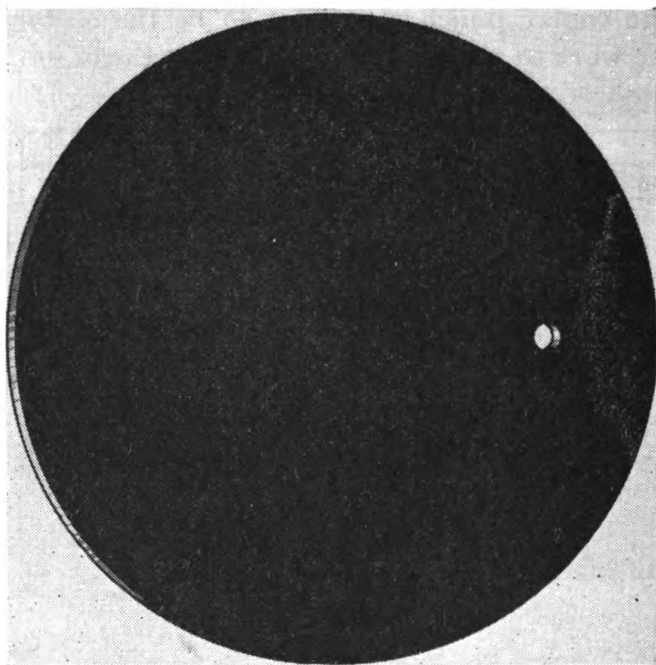


FIGURE 84.—Example of eccentric hole.

forged with lugs at the ends which can be centered for machining the crankpins. If such lugs are not provided, adapters for the shaft ends must be used which can be countersunk with the centers desired. In the upper part of figure 83, jigs have been bolted to the ends of the shaft and centered at points in line with the crankpin centers. To machine the crankpin, a cutting tool must be used which will reach between the crank arms.

22. Special set-ups.—*a. Faceplate work.*—Lathes are generally equipped with a small and a large faceplate (fig. 85). The small faceplate or driving plate ordinarily has one deep notch in its circumference and is used only for driving work held between centers as described in paragraph 16*f*. The purpose of the large faceplate is to provide a plane surface at right angles to the headstock spindle

to which work can be secured by T-bolts, straps, planer stops, angle plates, or whatever means its size and shape permit. The large faceplate is especially convenient for eccentric turning, drilling, or boring, and for holding many odd-shaped pieces which cannot be set up by any other method. It is impossible to describe in detail in this manual all the ways in which large faceplates can be used. Figures 86 and 87 show a few of its most common applications. The examples illustrated show that the uses of the large faceplate are limited only by the machinist's judgment and experience in setting-up operations.

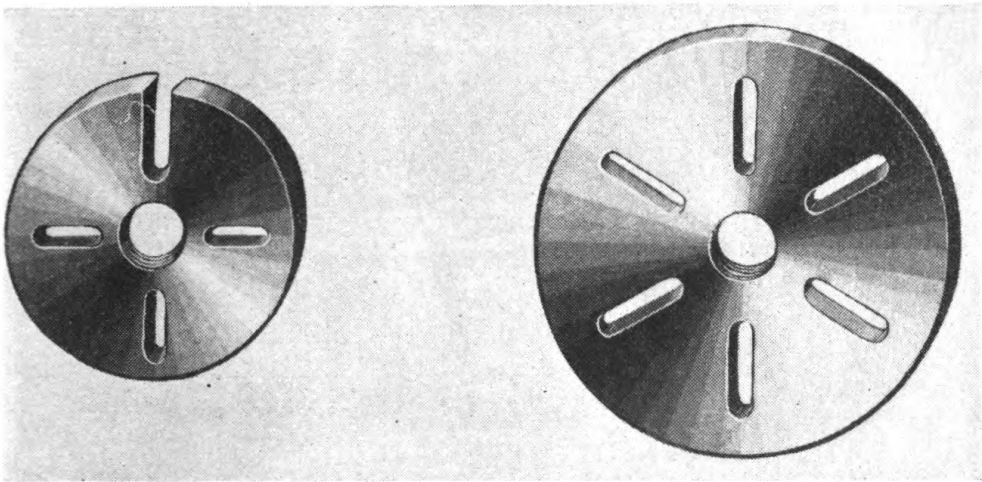


FIGURE 85.—Small and large faceplates.

b. Mandrels.—(1) Work which cannot be held between centers because its axis has been drilled or bored out and which is not suitable for holding in a chuck or on a faceplate, is usually machined on a mandrel which is a tapered axle pressed into the hole in the work to support it between centers. (A mandrel should not be confused with an arbor, which is a similar device, but used for holding tools rather than work.) Figure 88 shows a typical solid mandrel for holding work to be machined in the lathe.

(2) Mandrels are generally made of hardened steel, ground to a slight taper of from 0.0005 to 0.0006 inch per inch. They are provided with very accurately countersunk work centers; the ends are a little smaller than the body and have flats milled on them as a driving surface for the lathe dog. The size is always stamped on the *large end*. A good example of work held on a mandrel is shown in figure 89 in which a pulley has been set up on a mandrel between centers in order to turn its outer surface.

(3) To insert the mandrel into the work, first oil both the work and the mandrel, and drive the mandrel into the hole, small end first, with a soft hammer until the frictional resistance created by the taper is sufficient to drive the work. The mandrel is then set up between centers as though it were part of the work by the usual method described in paragraph 16f.

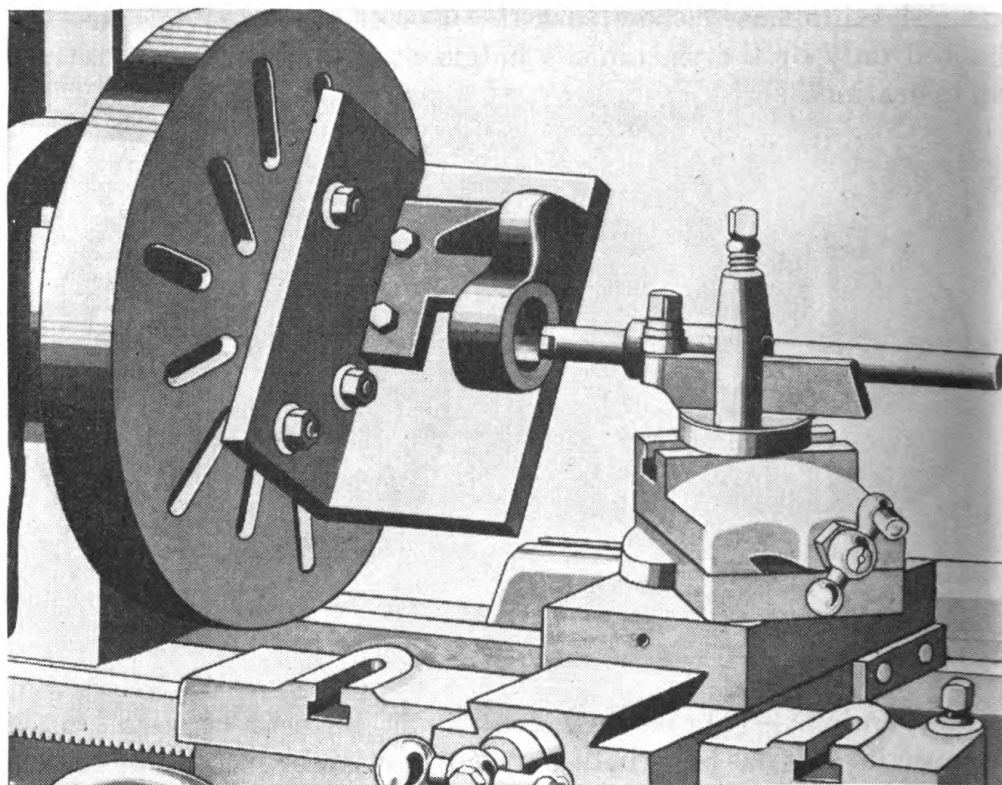


FIGURE 86.—Work held on faceplate for boring eccentric hole.

(4) Since solid mandrels are made to fit holes of some specific size and since they are subject to a certain amount of wear, an expanding mandrel as shown in figure 90 is often used for lathe work. An expanding mandrel is a chuck arranged so that the grips can be forced out against the interior of the hole in the work.

23. Boring.—*a. Tools.*—Boring is the process of enlarging and finishing a hole with a boring tool. The boring tool and inside threading tool are illustrated in figure 24 and described in paragraph 13b. Except for the method of holding the cutter bits, the procedure for boring is the same as for any external lathe operation.

b. Boring bars.—(1) Figure 91 shows a typical boring tool for use in the lathe. The bar is held in the tool post and the cutter bit secured to the other end of the bar. The attachments allow the

tool to be set at a 90° , 45° , or 30° angle for boring, facing, or threading. The tool can then be fed longitudinally or crosswise by means of the usual controls. Work to be bored must be set up in

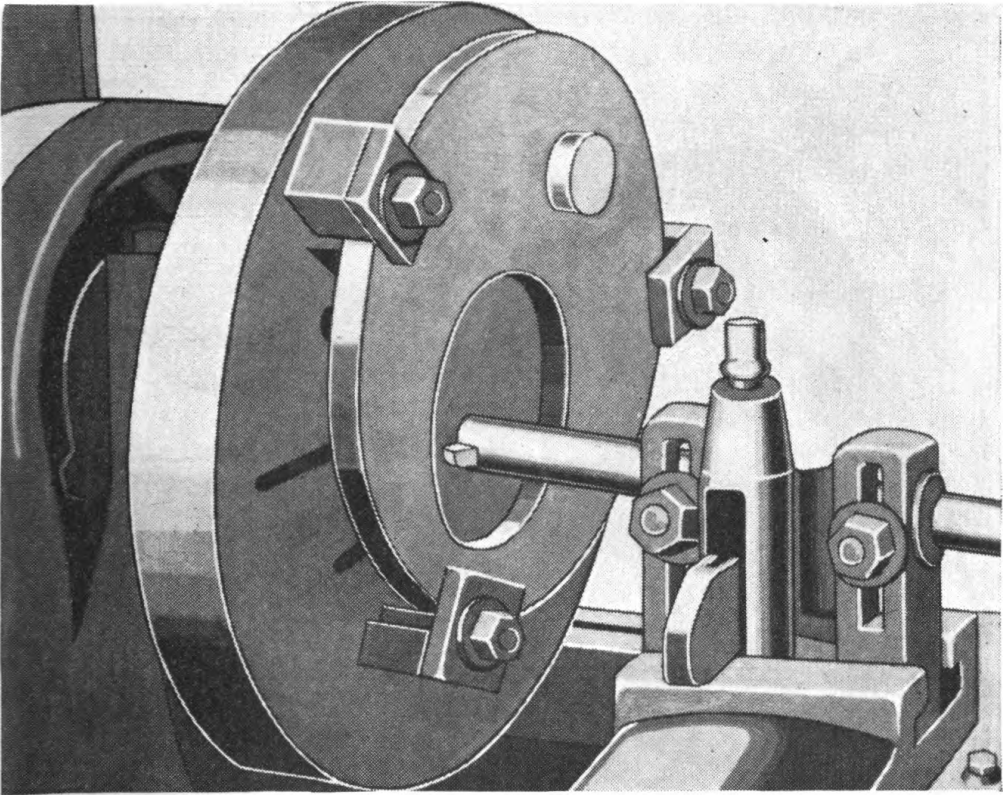


FIGURE 87.—Bracket held for boring and facing on angle plate and faceplate.

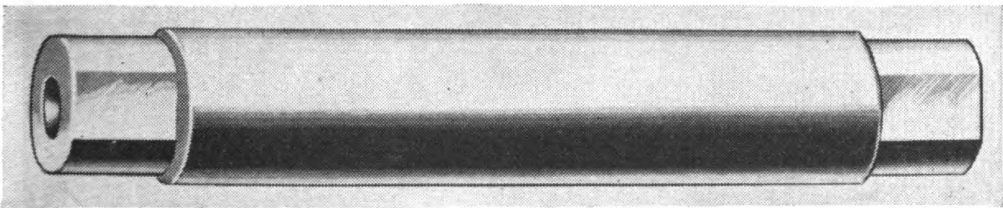


FIGURE 88.—Solid lathe mandrel.

a chuck or on a faceplate. Taper boring is practicable either by means of the compound rest, as described in paragraph 18*b*, or by means of a taper attachment.

(2) Another method of boring in the lathe is to set up the work on the lathe carriage and bore it by means of a cutter bit secured in a boring bar held and turned between the lathe centers. This method is generally suitable only to straight bores or very slight tapers.

24. Special lathe work.—*a. Knurling.*—Knurling is the process of embossing the surface of a piece of work to give it a gripping surface which has a finished appearance. The operation is similar to

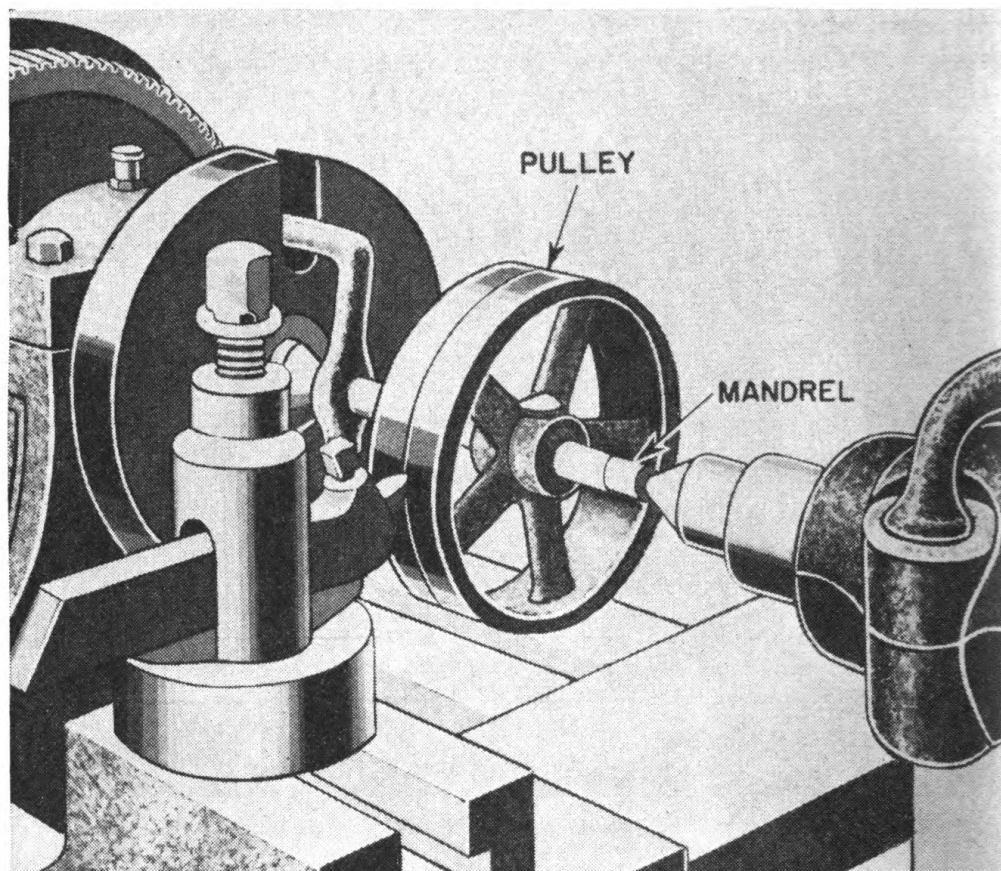


FIGURE 89.—Turning a pulley on a mandrel.

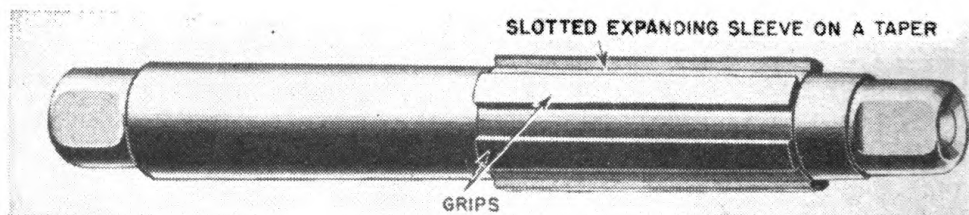


FIGURE 90.—Expanding mandrel.

straight turning, except that a knurling tool (fig. 92) is secured in the tool post instead of the usual cutting tool.

b. Filing and polishing.—Tool marks may be removed from turned work by filing and polishing the surface in the lathe. A fine-cut mill file gives best results; the spindle speed should be fast enough

so that the work makes two or three revolutions for each stroke of the file. Caution is needed to avoid filing a piece too much, which will throw the work out of round. A very bright finish can be obtained by following the filing with a polishing operation in which a piece of oiled emery cloth is held between the file and the rotating work.

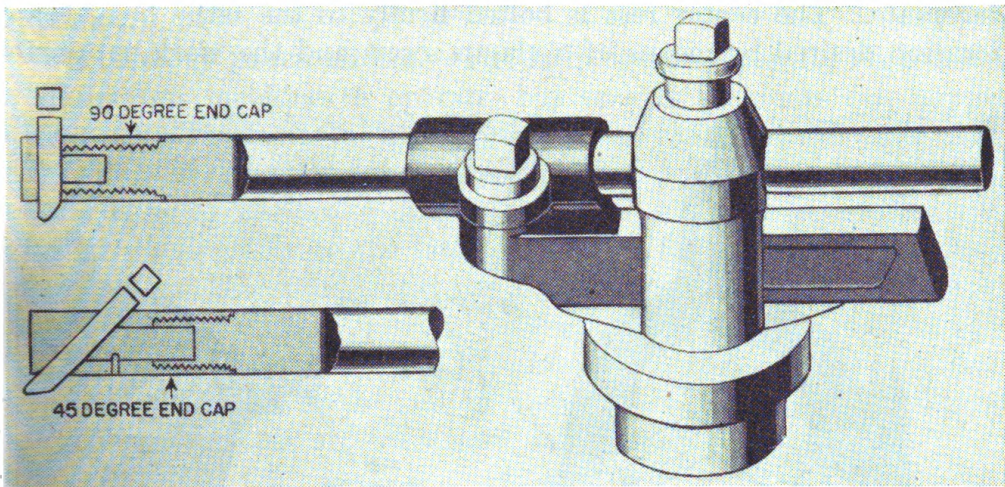


FIGURE 91.—Typical boring tool.

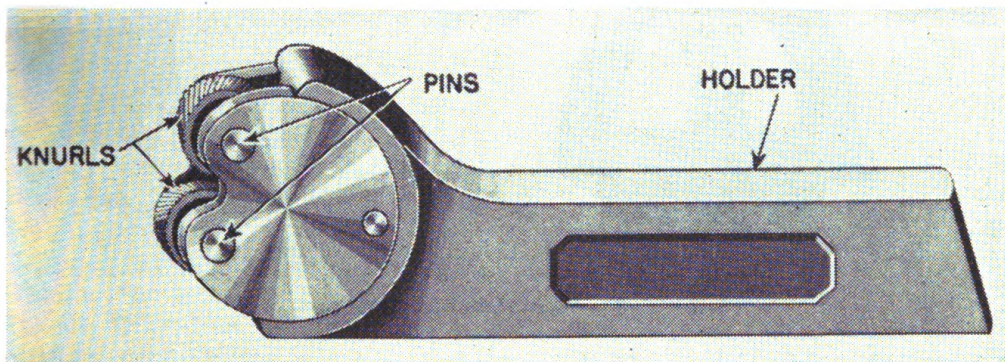


FIGURE 92.—Knurling tool.

c. Lapping.—Paragraph 9*d* describes the process of lapping holes by means of a drill press. The same operation can be done conveniently in the lathe. The set-up is similar to drilling in the lathe (par. 19*a*) except that a lap is substituted for a twist drill.

d. Spring and coil winding.—The range of longitudinal power feeds available on most lathes makes them convenient for winding electrical coils of all kinds. Special gearing and wire guides are available which make the operation easier than the usual methods of winding by hand. Coil springs can be wound on a mandrel by

means of a wire guide. The coils are spaced uniformly by using the lead screw and half nuts as when cutting threads.

e. Use of center and follower rests.—(1) A center or steady rest and a follower rest (fig. 93) are usually included with a standard lathe equipment. The center rest is used for turning long shafts of small diameter and for holding work for boring or internal threading when the work projects a considerable distance from the chuck or faceplate. The center rest is bolted firmly to the lathe bed at any location desired by means of a clamp screw and the work supported

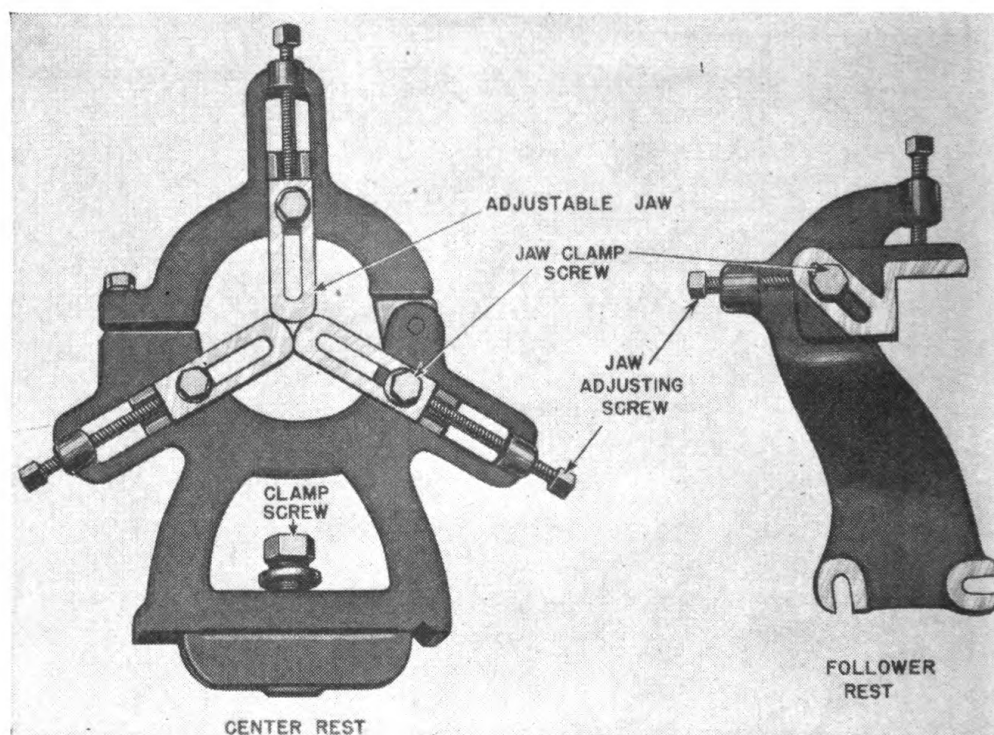


FIGURE 93.—Center and follower rests.

by the adjustable jaws. If the work is rough or unfinished, a portion of it must be turned to a finish which is supported in the rest jaws. The jaws must be kept oiled during use.

(2) The follower rest (fig. 93) is similar to the center rest in that it is used for supporting long, slender shafts while they are being turned or threaded; it usually has only two jaws and is fastened to the lathe carriage so that it follows the cutting tool and supports the work close to the cut at all times. The cut must be started and continued for a longitudinal distance a little greater than the width of the jaws before the follower rest is placed against the work.

f. Milling attachment.—Milling attachments are available for most lathes. The subject of milling is covered in detail in section IV. When the operator has become familiar with a milling attachment, he should be able to perform various milling operations with it.

g. Grinding attachments.—Section VI describes various shop grinding operations in detail. A number of attachments are available for the lathe which converts it into a grinder for work within its capacity.

25. Safety precautions.—The lathe is not generally regarded as a dangerous machine to operate, but serious accidents can happen unless some fundamental safety precautions are observed while using it. The general safety precautions for the machinist (par. 1*b*) apply to the lathe as well as to other machines. A safety precaution that applies particularly to the lathe is: do not file work while the back gears are in; the spindle speed is usually too slow to keep the work sound and the work is moving with too much force to stall if anything gets caught in it. All operators of machine tools should have their sleeves rolled up or cut off above the elbow. Loose fitting or ragged shop clothes are extremely dangerous.

SECTION IV

MILLING MACHINE

	Paragraph
General.....	26
Parts and nomenclature.....	27
Controls.....	28
Milling cutters.....	29
Kinds of milling cutters.....	30
Holding the cutter.....	31
Grinding milling cutters.....	32
Holding the work.....	33
Speeds, feeds, and coolants.....	34
Milling operations.....	35
Indexing.....	36
Operating and safety precautions.....	37

26. General.—*a. Principle of milling machine.*—The milling machine is used for machining flat, curved, or irregular surfaces of various kinds. Work is fed against one or more rotating cutting tools called milling cutters, each of which has a number of cutting edges called teeth. This principle is exactly opposed to that of the lathe (sec. III) which removes metal by a single-edged cutting tool fed into the rotating work.

b. Types of milling machines.—Milling machines fall into two main groups: those capable of performing a variety of operations

(general purpose type); and those restricted by their design to the performance of some specific operation, such as gear cutting, bolt head milling, etc. The general purpose machines are broadly classified into three groups: the column and knee type; the manufacturing type; and the planer type. Column and knee machines, which are in most common use, are made with either horizontal or vertical spindles, plain or universal tables, and for either floor or bench mounting. This manual explains in detail the principles and operation of the universal horizontal-spindle machine which will perform all common milling operations and which is generally used in Army shops. Milling machine sizes are specified by arbitrary numbers which each manufacturer has adopted as his own standard.

c. Range of operations.—It is not possible to name every operation within the range of milling machines. Generally they are excellent for forming flat surfaces or irregular profiles; cutting splines, slots, or keyways; accurate drilling; gear cutting; spiral milling, etc. As a rule they are furnished with a dividing head for indexing operations; other attachments are available for vertical spindle work, circular milling, slotting, taper milling, and other special jobs.

27. Parts and nomenclature.—*a.* Figure 94 illustrates a popular universal horizontal-spindle milling machine with its various parts clearly shown. The machines of various manufacturers differ in refinements of detail but the essential parts and fundamental principles are the same in all. Whenever possible, study the manufacturer's manual dealing with the make of machine being used.

b. The base and column are cast in one piece and support the entire machine. The column contains the motor (if the machine is driven by an individual motor and almost all Army milling machines are), the coolant pump if one is provided, and the various gears and controls which regulate the available spindle speeds and transmit power for the automatic table feeds. The knee slides vertically on guides machined into the front of the column and the overhead arm or overarm moves horizontally on dovetail slides machined on top of the column. The base catches any overflow of coolant from the table and returns it through a drain to the pump or other source by gravity.

c. The knee supports the saddle and contains the gears and controls which regulate the hand and power table feeds. It is moved vertically on its guides by the elevating screw or vertical feed handle. The saddle moves on dovetail slides on top of the knee.

d. The saddle supports the table. It moves parallel to the spindle on the dovetail slides machined on the knee and has dovetail slides

machined in its top on which the table moves. Plain milling machine saddles limit the table movement at right angles to the axis of the spindle. Universal machine saddles similar to the one illustrated can be set at any angle up to, usually, 45° to the axis of the spindle in a

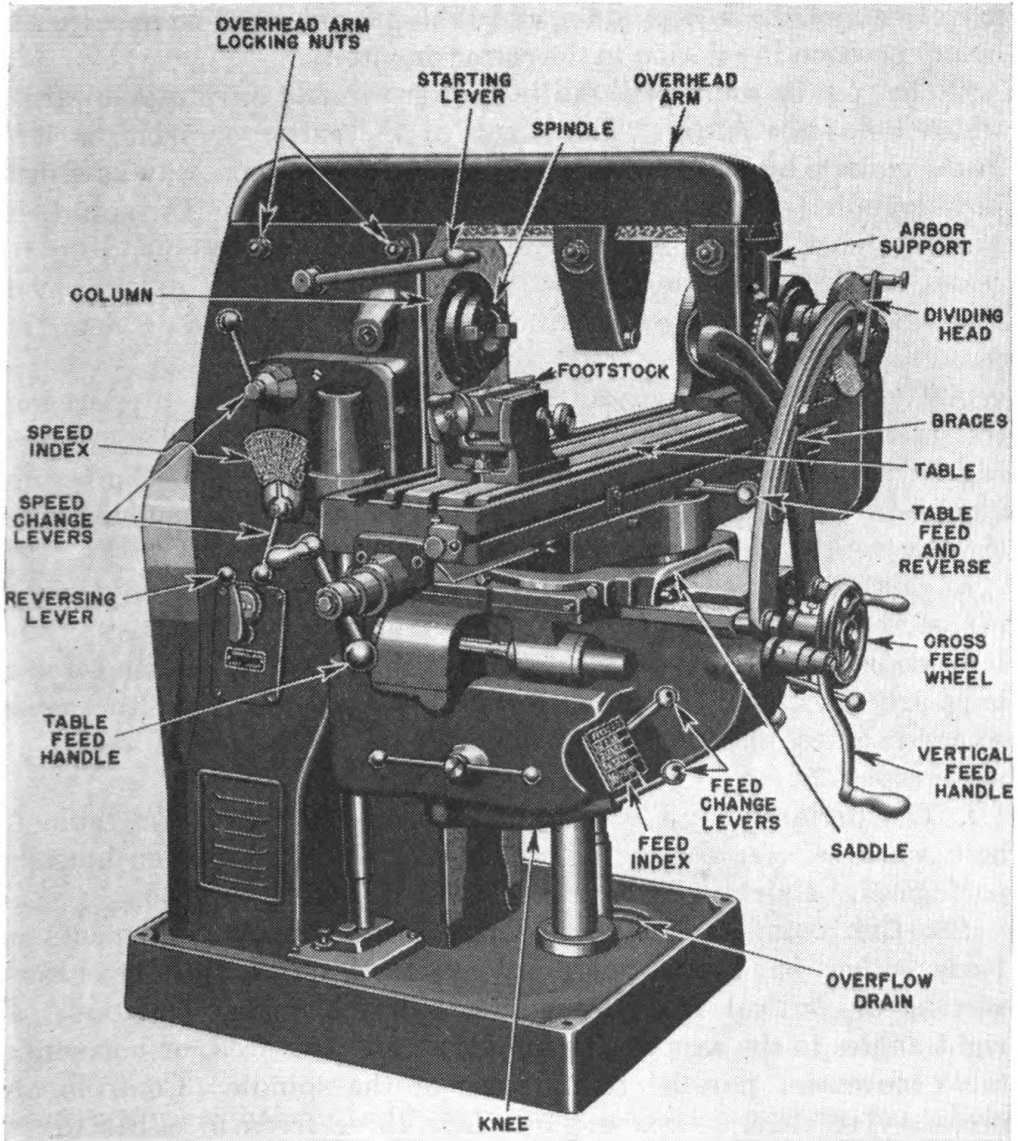


FIGURE 94.—Universal milling machine.

horizontal plane. The swivel base of a universal saddle is graduated in degrees so accurate angular settings can be obtained.

e. The table is the surface to which the work is secured and by which it is fed into the milling cutter or cutters while being machined. It is carefully ground level and provided with T-slots for clamping

the work or a work-holding device to it. These T-slots run exactly parallel to the table's direction of travel. The table moves horizontally in the dovetail slides on top of the saddle. (Tilting tables are a special attachment which provide a vertical angular setting for milling tapers.) The combined horizontal, vertical, and angular movements of the knee, saddle, and table permit work to be adjusted in any position in relation to the cutter or cutters.

f. The spindle rotates the milling cutter directly or rotates an arbor which holds the cutter. The shank of the cutter or arbor is held in the spindle by means of a taper and slot in the same way as a drill press spindle (par. 7a) holds and rotates a twist drill. The spindle is also provided with lugs or recesses that fit into corresponding recesses or lugs in some arbors to furnish a positive driving connection. The methods of holding cutters are discussed in detail in paragraph 31.

g. The overarm or overhead arm holds the outer arbor support and the intermediate arbor support, if one is used, or both. The overarm can be moved horizontally parallel to the spindle and clamped in any position within its range by the clamping screws or removed from the machine entirely for some operations.

h. The arbor supports provide bearing surfaces for the outer end of an arbor which holds and rotates some types of cutters. The intermediate arbor support furnishes an additional bearing for a long arbor, especially where two cutters are mounted on the arbor some distance apart. These supports are explained in detail in paragraph 31.

i. The dividing head and footstock are clamped to the table to hold work between centers on which an indexing operation must be performed. Indexing is described in detail in paragraph 36.

28. Controls.—*a. General.*—Three distinct table movements or feeds, either hand or power, are available on milling machines: vertical or vertical feed; longitudinal or table feed, horizontal at right angles to the axis of the spindle; transverse feed, or horizontal table movement parallel to the axis of the spindle. Controls are necessary to engage or disengage these three feeds in either direction; most milling machines have additional "trip" controls which automatically disengage the power feeds at some point set by the operator. Figure 95 illustrates a type of bench milling machine used for Army instruction with its various controls clearly shown. The location and operation of these controls vary somewhat with different manufacturers; but the same principles apply to all machines, and each manufacturer's operating manual will clarify any diffi-

culties. The machinist should thoroughly understand the controls illustrated in figure 95 and not attempt to operate a milling machine until entirely familiar with them.

(1) *Arbor support arm clamps.*—The arbor support arm is clamped tightly in position by the two arbor support clamps shown in figure 95. The arm can easily be removed from the machine by

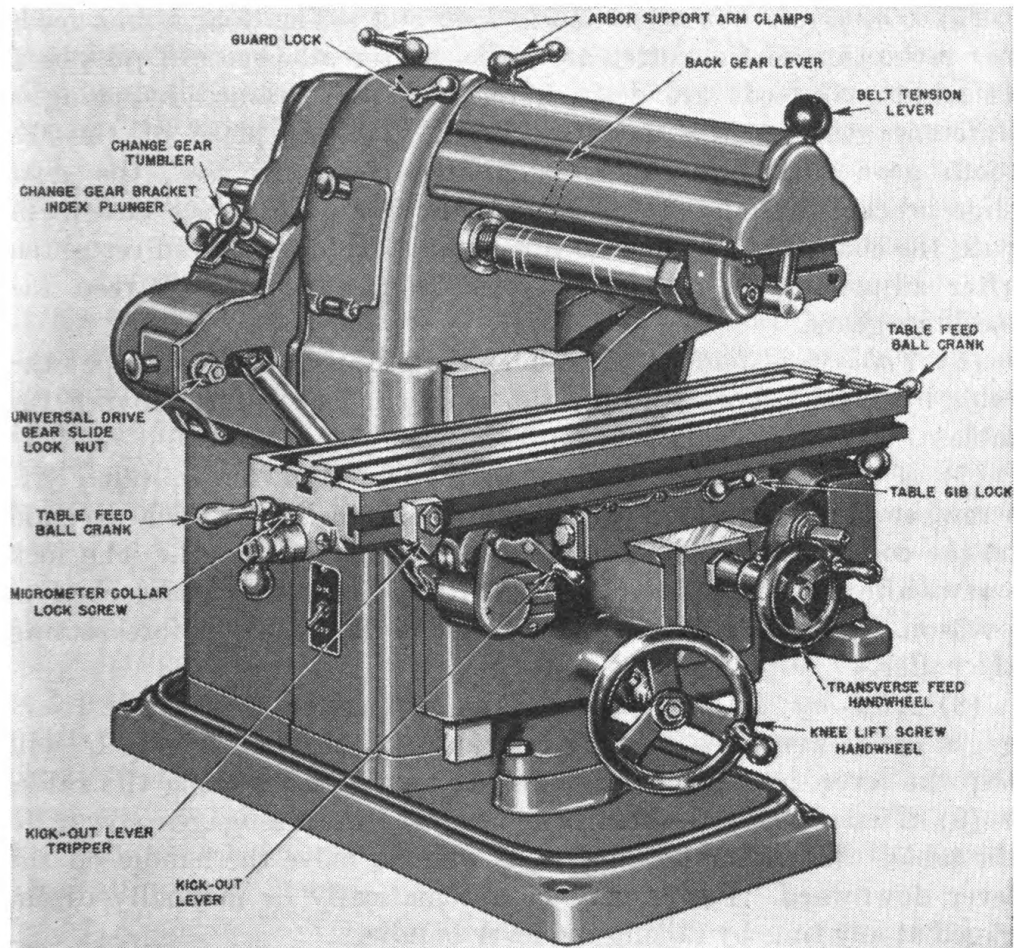


FIGURE 95.—Bench type plain milling machine.

loosening these handles and the guard lock and pulling the whole unit forward.

(2) *Guard lock.*—Locks cutter guard in position.

(3) *Change gear tumbler.*—This control has three positions. In the center one it places the tumbler gears in neutral, that is, not in mesh with the spindle gear. In the other two positions the gears are in forward and reverse, depending upon which tumbler gear is in mesh with the drive toggle gear.

(4) *Change gear bracket index plunger.*—This plunger sets the change-gear bracket at four different positions. Pull out the plunger and rotate the change-gear bracket by the change-gear bracket knob (not shown) until the plunger clicks into place. This knob rotates the change-gear bracket for any one of the following four feeds: 0.003 inch; 0.006 inch; 0.0125 inch; 0.025 inch. These feeds are explained in detail in *d* below.

(5) *Universal drive gear slide lock nut.*—The four table feeds per revolution of the cutter are sufficient for all practical purposes. If additional feeds are desired, they can be obtained by using a different gear on the universal drive shaft in place of the 64-tooth gear furnished. When a different gear is used, the gear slide bracket must be moved so that the new gear comes into mesh with the change gears. To do this, loosen the lock nut and retighten after adjustment is made. Leave sufficient clearance between the meshing gears.

(6) *Table feed ball cranks.*—These cranks are used to propel the table by hand. Always loosen the table gib lock before moving the table.

(7) *Micrometer collar lock screws.*—The feed screw, table lift screw, and cross slide screw have micrometer collars. Each division on the collar represents one thousandth of an inch travel. The lock screws allow the collars to be set at the zero reading and locked in position. Always remove all backlash in the screw before setting the collar.

(8) *Kick-out lever tripper.*—This tripper automatically disengages the power table feed by tripping the kick-out lever. It will trip the lever from either direction at any position along the table.

(9) *Kick-out lever.*—This lever engages or disengages power to the table. Whenever power feed is wanted, move the handle on the lever downward. Power may be automatically or manually disengaged at any time by raising the lever handle.

Caution: Be sure to disengage the automatic table feed before the table reaches its limit of travel. The machine may be very seriously damaged if the table is allowed to jam against the cross slide.

(10) *Knee lift screw handwheel.*—Raises or lowers the knee table assembly. Always loosen the gib lock located on the right side of the knee before moving the table.

(11) *Transverse feed handwheel.*—Moves the table forward or backward. Always loosen the gib lock located on the right side of the cross slide before turning the wheel.

(12) *Belt tension lever.*—By moving the belt tension lever to the right, the belts are tightened for operation. If belt tension becomes slack, tighten as described in *b* below.

(13) *Back gear lever.*—To put the back gears into mesh, always pull out the grooved pin which engages the large spindle gear with the spindle pulley. This can readily be done with a screw driver or similar tool. The spindle gear is accessible through the left side of the head after pulling open the side cover plate. Move the back gear lever toward the left until the back gears click into position. To disengage the back gears, move the back gear lever to the right. Then engage the spindle gear pin into the pulley by pushing against the end of the pin with a screw driver and at the same time turning the spindle pulley by hand until the pin drops into place.

b. Adjustments.—(1) *Spindle adjustment.*—The tapered roller bearing does not need frequent adjustment. If the spindle spins too freely or play is noticeable when the spindle is pushed back and forth, adjust the headstock bearings as follows: Run the miller between 30 minutes and an hour to warm up the spindle (a temperature rise of 50° F. increases the length of the spindle about 0.002 inch between bearings). Remove the change-gear guard on rear of miller to get at the thrust nut by loosening the capscrews. Then loosen the setscrew on the thrust nut at the extreme left end of the spindle and turn it up until no play can be detected in the spindle. Advance this thrust nut $\frac{1}{32}$ turn (equal to one tooth of the spindle gear) past that point in order to provide the correct preload. Tighten the setscrew.

(2) *Belt tension.*—(a) *Spindle belt.*—Two headless setscrews located on the countershaft hanger and bearing against the rockershaft serve to take up the belt slack. When adjusting these screws, put the belt tension lever in the tension position and relock the screws after adjustment is made.

(b) *Motor belt.*—Tension on this belt is decreased or increased by moving the motor base up or down by turning the two “hex” nuts located on the stud which goes through the upright on the motor base. After the adjustment is made, lock the nuts in place.

NOTE.—In either case do not have belt tension too tight. Excessive belt tension will cause unnecessary bearing wear.

(3) *Table and cross slide.*—It is very important that the gibs on the table and cross slide always fit snugly and that there is no play. Gib adjusting screws are provided for making any necessary adjustment. The gib screws should always be locked in place with the nuts

after adjustment is made. The table and cross slide should move with a slight "drag" effect. Any play will cause chattering cuts and rough finishes.

(4) *Knee*.—The knee gib should also fit snugly at all times but not tightly enough to hinder movement of the knee. Always lock the setscrews in place after adjustment is made. No play should ever be present in the knee slides. Shims between the knee and knee anchor plates afford necessary take-up adjustment. The shims are 0.010 inch thick, made up of four 0.002 and two 0.001 inch leaves held together with a light film of solder. They can easily be separated with a knife blade. The 0.001 inch side is colored gray.

c. Spindle-speed chart.—A speed chart located on the side cover plate on the head of the miller shows all the speeds available and the belt set-ups to obtain them.

d. Reading feed per revolution dial.—Four different table feeds per revolution of the cutter are available. These feeds are shown in each case under the title "Feed per rev." After the proper table feed has been decided upon, set the dial so that the nearest approximate table "Feed per rev." appears through the dial window. Below the "Feed per rev." are two columns, one showing the rpm and the other the feed in inches per minute. After the proper cutter speed has been obtained in rpm, as explained in paragraph 34a, refer to the rpm column and choose the nearest approximate rpm shown. The adjacent number in the feed column will show the distance the table will travel in inches per minute at that particular spindle speed. By dividing this figure into the length of the portion to be milled, the time required for milling can easily be calculated. There is a total of 32 different table feeds per minute on the "Feed per rev." dial. Do not confuse "Table feed per rev. of cutter" with "Table feed per minute."

Caution: When the teeth of the cutter come in contact with the work, they should always rotate against the direction in which the work is traveling; or the machine may be seriously damaged.

29. Milling cutters.—*a. General characteristics*.—As paragraph 26a explained, milling is the process of removing metal by means of a rotating cutting tool or tools having a number of cutting edges called teeth. Such tools are known as milling cutters or mills. They are usually made of high-speed steel and are available in a great variety of shapes and sizes for various purposes. (There are over 50 kinds of cutters in general use, over 4,000 stock sizes.) The names of the most common classifications of cutters, their uses, and in a general way the sizes best suited to the work in hand should be known. Figure

96 shows a side view and cross section of a common milling cutter with its parts indicated. These parts in some form are common to all types of cutters.

b. Types of teeth.—The teeth of milling cutters are either right-hand or left-hand, viewed from the back of the machine. Right-hand mills cut when rotated clockwise; left-hand mills cut when rotated counterclockwise.

(1) Saw teeth similar to those shown in figure 96 are generally either straight or helical in the smaller sizes of plain mills, slitting

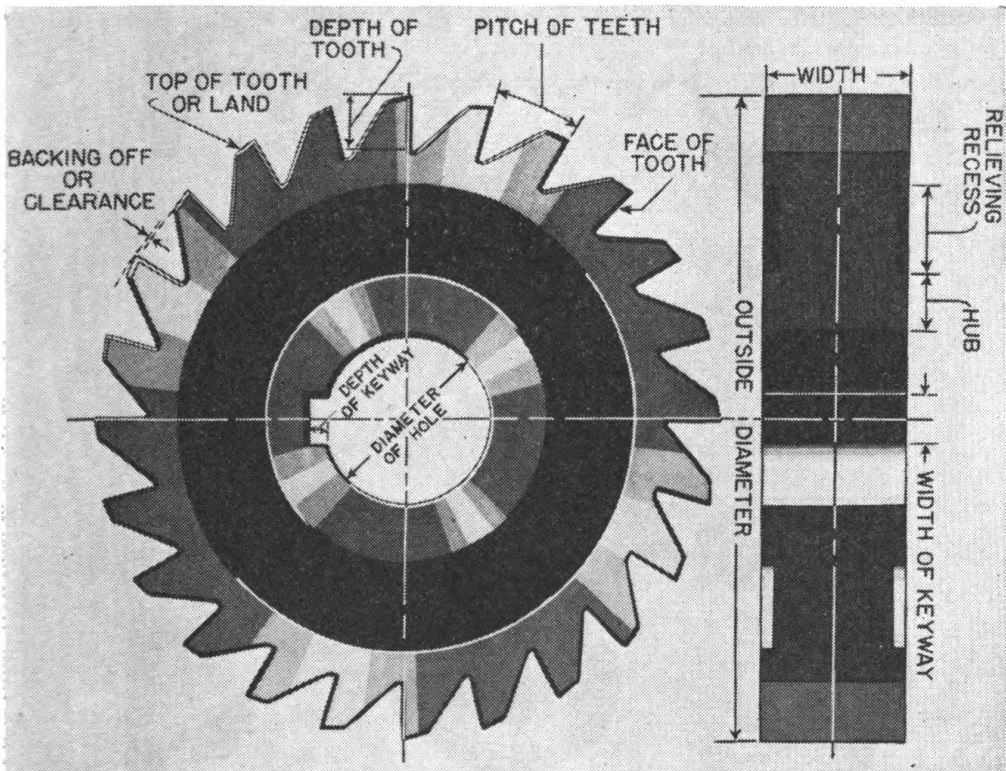


FIGURE 96.—Parts of milling cutters.

cutters, and end mills. The cutting edge is given about 5° clearance. Sometimes the teeth are provided with offset nicks which break up the chips and make coarser feeds possible.

(2) Formed teeth (fig. 97) are usually specially made for machining irregular surfaces or profiles. The possible varieties of formed-tooth mills are almost unlimited. Formed cutters are sharpened by grinding the faces of the teeth radially and repeated sharpenings are possible without changing the contour of the cutting edge.

(3) Inserted teeth (fig. 98) are blades of high-speed steel inserted and rigidly held in a blank of machine steel or cast iron. Different

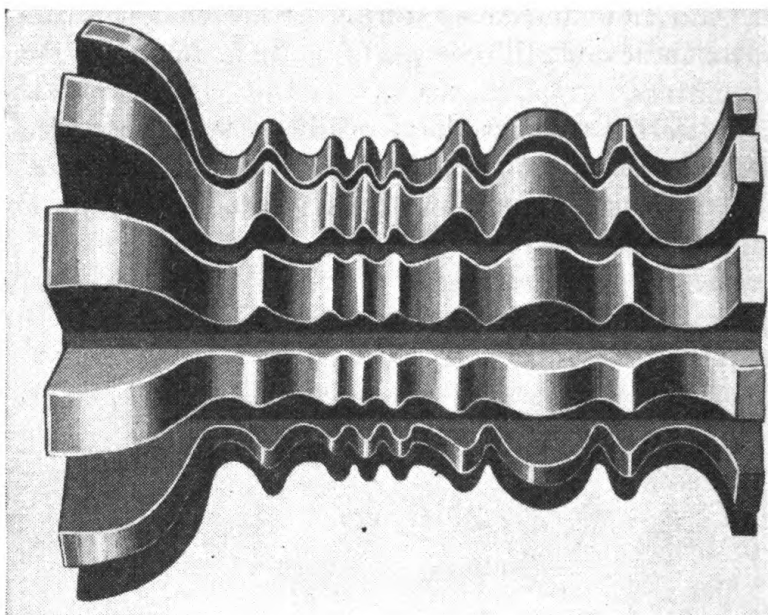


FIGURE 97.—Formed milling cutter.

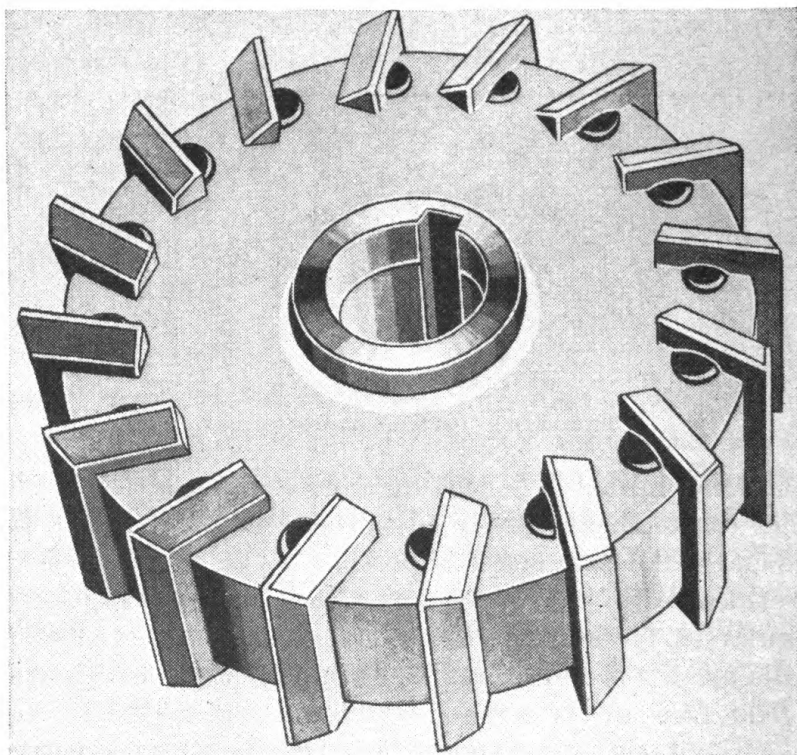


FIGURE 98.—Inserted tooth cutter.

manufacturers use different methods of holding the blades in place. Inserted teeth are especially economical and convenient for large-sized cutters because of their reasonable initial cost and because worn-out or broken blades can be replaced easily.

30. Kinds of milling cutters.—*a. Plain mills.*—The most common form of milling cutter is known as a plain mill. It is merely a cylinder having teeth cut on its periphery for producing a flat horizontal surface (or a flat vertical surface in the case of a vertical spindle machine) as shown in figure 96. When the cutter is over $\frac{3}{4}$ inch wide, the teeth are usually helical, which gives the tool a shearing action and requires less power, reduces chatter, and produces a

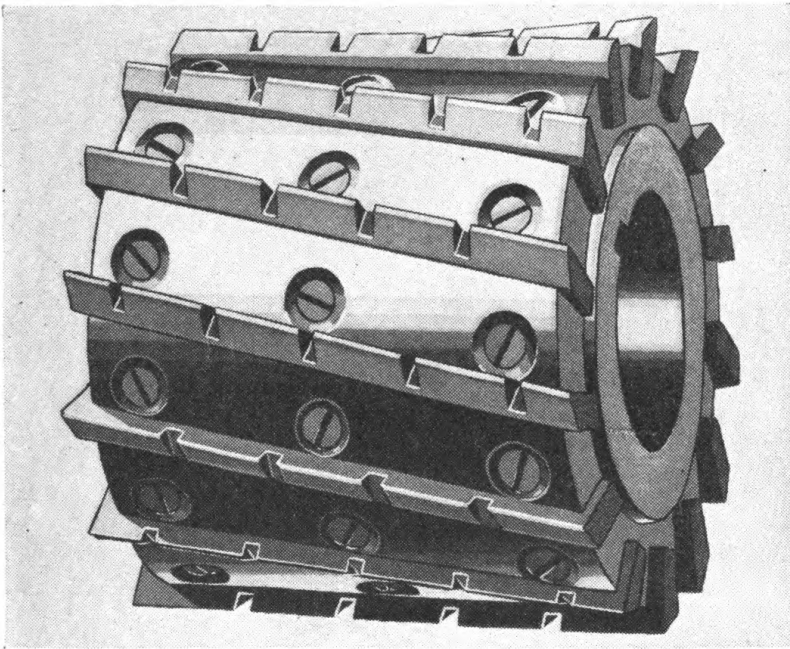


FIGURE 99.—Slabbing cutter with helical nicked teeth.

smoother finish. Cutters with faces less than $\frac{3}{4}$ inch wide are sometimes made with staggered or alternate right- and left-hand helical teeth. The shearing action, alternately right and left, eliminates side thrust on the cutter and arbor. When a plain cutter is considerably wider than the diameter, it is called a slabbing cutter; slabbing cutters generally have nicked teeth as shown in figure 99.

b. Slitting mills.—Figure 100 illustrates a slitting mill which is essentially a very thin plain mill. It is ground slightly thinner toward the center to provide side clearance and is used for metal sawing and for cutting narrow slots.

c. Side mills.—Figure 101 shows two forms of side milling cutters. Side mills are essentially plain mills with the addition of teeth on one or both sides. (When teeth are added on one side only the cutter is called a half side mill.) Thus the tool cuts on the periphery and one or both sides at the same time. They are ordinarily used for machining slots. Interlocking side mills are used for forming comparatively wide slots with accuracy; their teeth are formed to interlock when mounted side by side. Interlocking side mills can be repeatedly

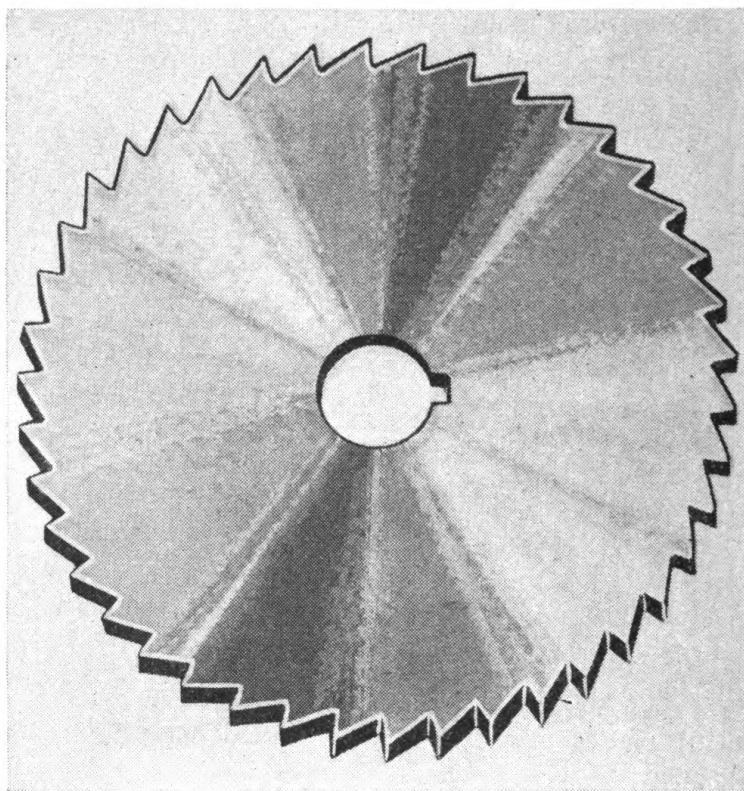


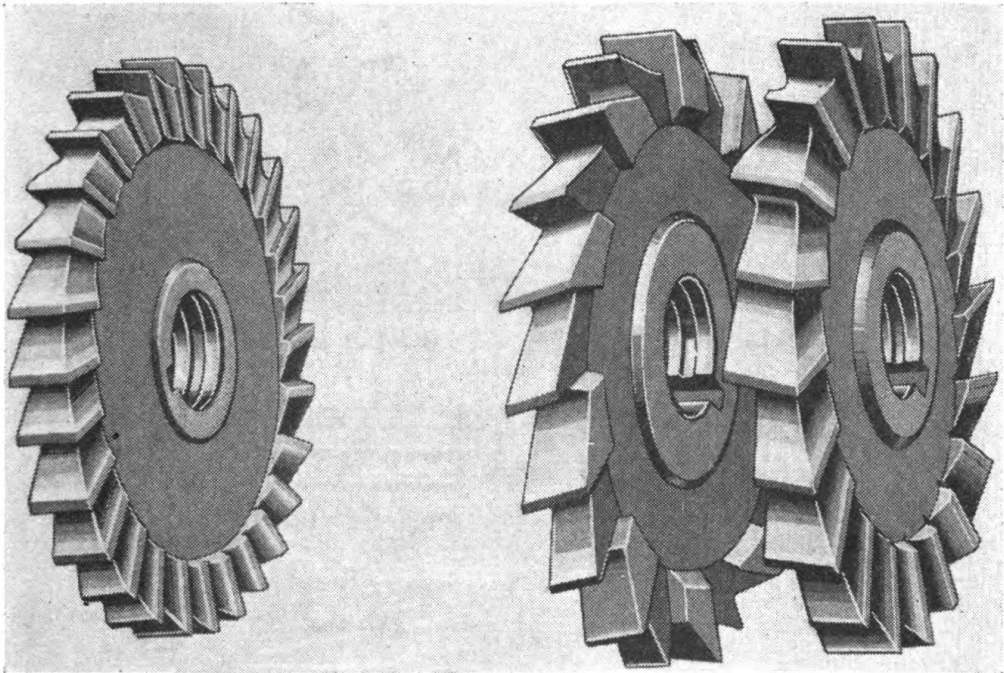
FIGURE 100.—Sifting mill.

sharpened without changing the width of slot they will machine. After each sharpening, put a washer between them wide enough to make up for the ground-off metal; the interlocking prevents the cutters from leaving a ridge on the work. When two side mills are used with a spacing collar between them to machine two parallel surfaces at once, they are known as straddle mills.

d. End mills.—(1) End mills have teeth on the end as well as the periphery. Six common types of end mills are illustrated in figure 102. They are used for a large variety of light milling operations such as machining the edges of fairly thin pieces, for squaring the

ends of small pieces, for making a shoulder where a fillet is desired, and for many slot and keyway cutting operations. End mills over $\frac{5}{8}$ inch in diameter usually have helical teeth on the periphery. The cotter mill is used for milling slots or keyways in solid metal where no drilled hole is provided for starting the cut.

(2) End mills over 2 inches in diameter are generally made separate from the shank as shown in figure 103 and when so made are called shell end mills. A shell end mill has a standard sized hole of proper diameter and a slot milled across the back to fit a tongue on



① Plain.

② Interlocking.

FIGURE 101.—Side mills.

the arbor which holds and rotates it. The cutter may be held on the arbor either by means of a cap screw or a shrink fit. To shrink on a shell mill, warm it in hot water, place it on the arbor tight against the shoulder, and cool the assembly by slowly immersing it in cold water *shank end first*.

e. Facing cutters.—Milling cutters with larger diameters and teeth on the end have no shanks but are fastened on the end of the milling machine spindle. They are then called face mills and are generally made with inserted teeth.

f. T-slot cutters.—Figure 104 illustrates a T-slot cutter used for cutting T-slots in machine work tables, etc. The central groove is

first milled with a side or end mill and the wider part milled with the T-slot cutter.

g. Angular cutters.—The peripheral teeth of an angular milling cutter are neither parallel nor perpendicular to its axis but at some

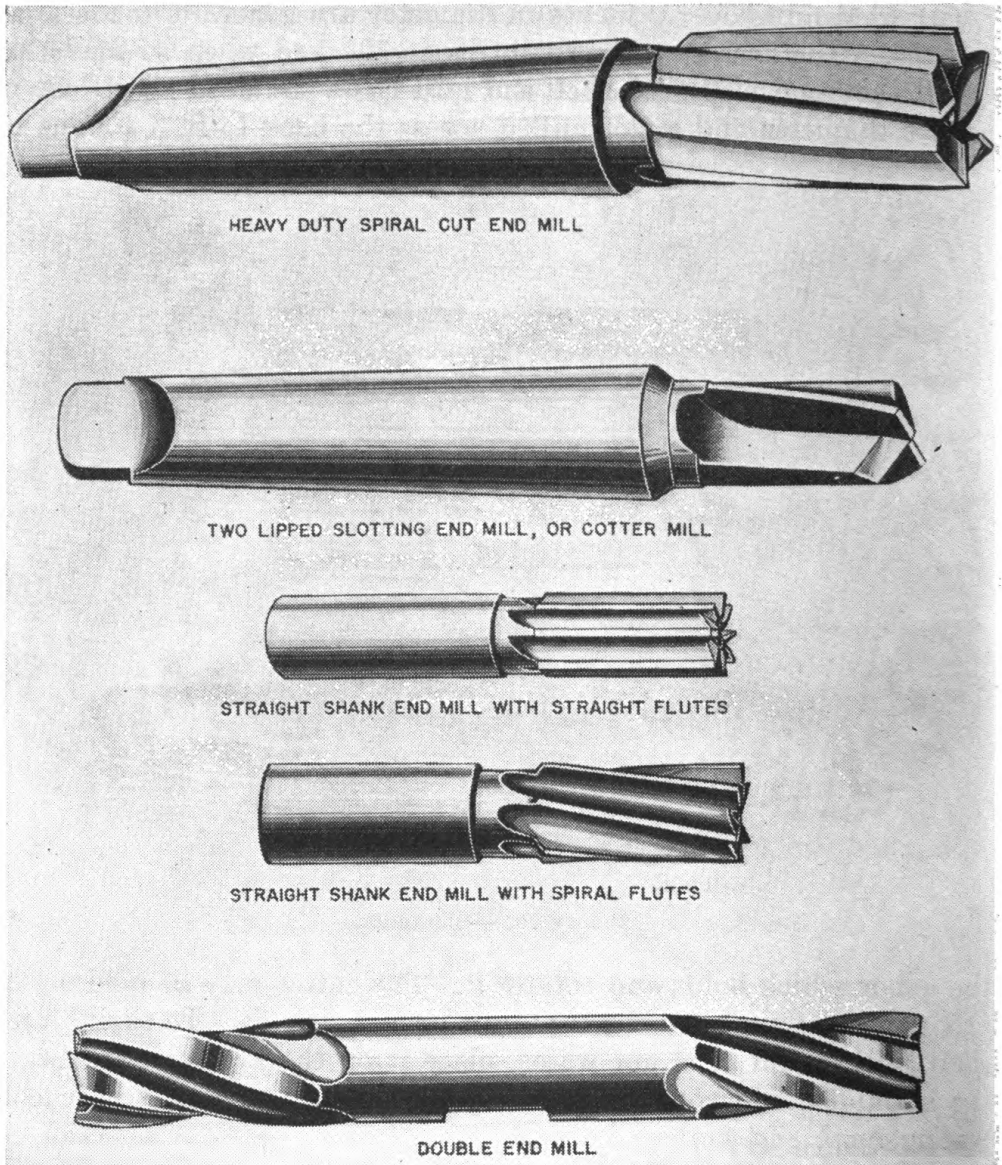


FIGURE 102.—Common types of end mills.

other angle such as 60° , 70° , or 80° . They are commonly used for cutting teeth in ratchet wheels, milling dovetails, etc. Sometimes side teeth are provided as shown in figure 105 to give a better finish to the side of the groove. Figure 105 also illustrates a double angle cutter which is used primarily for helical milling. These are usually

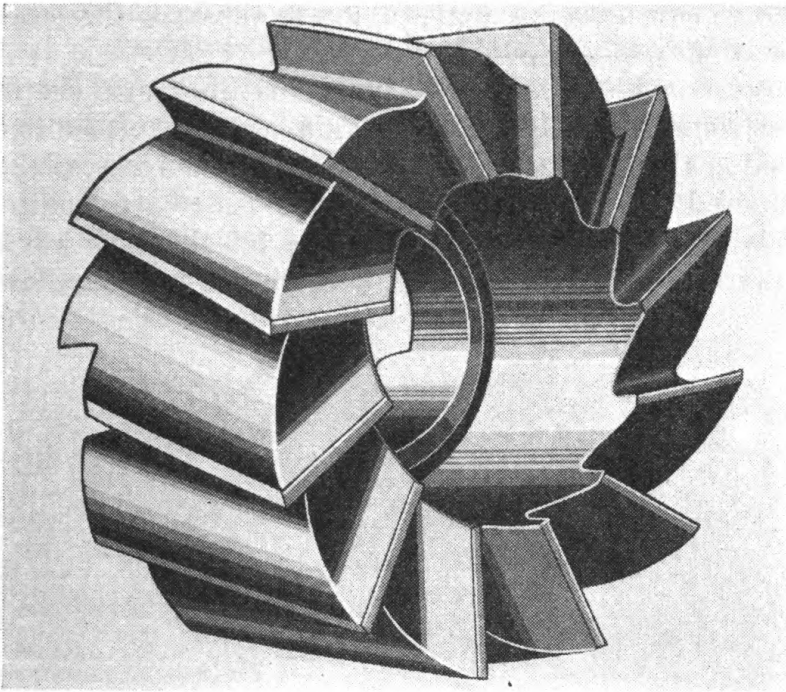


FIGURE 103.—Shell end mill.

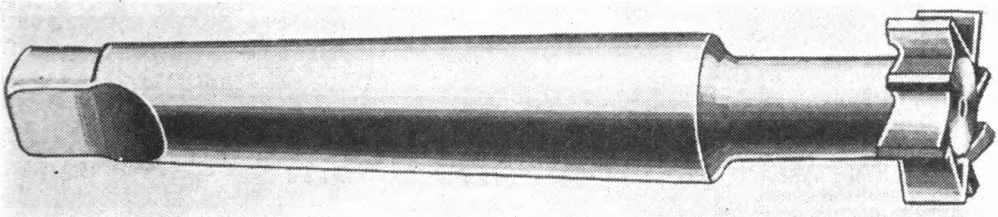
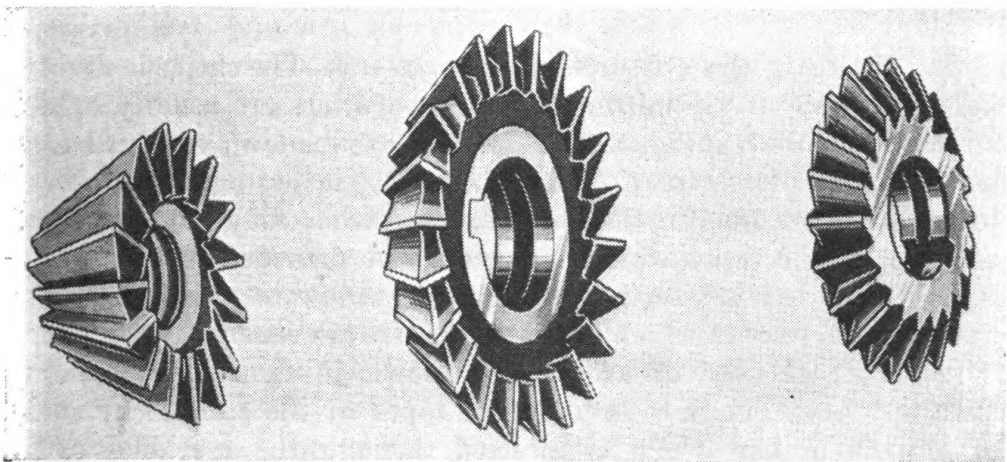


FIGURE 104.—T-slot cutter.



① Single with threaded hole.

② Single.

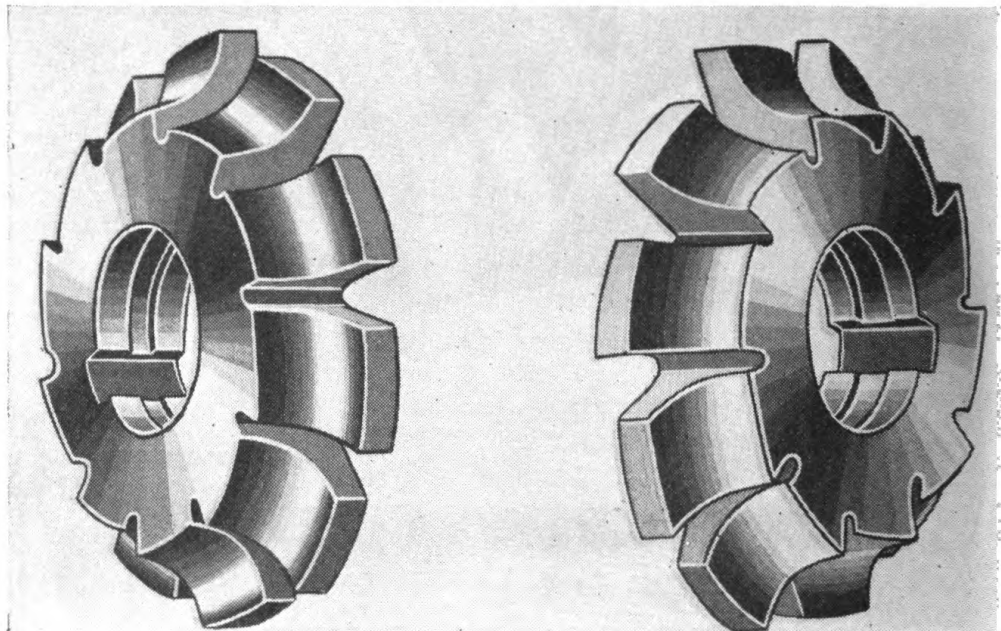
③ Double.

FIGURE 105.—Angular milling cutters.

made with one side at an angle of 12° to the axis, the other side at an angle of 40° , 48° , or 50° .

h. Corner rounding cutters.—Figure 106 illustrates the two types of corner rounding cutters. They are a style of formed cutter used for finishing the corners and edges of work and are available in any radius desired.

i. Convex and concave cutters.—Figure 107 shows convex and concave cutters used for machining half circles and smaller arcs of circles.



① Left hand.

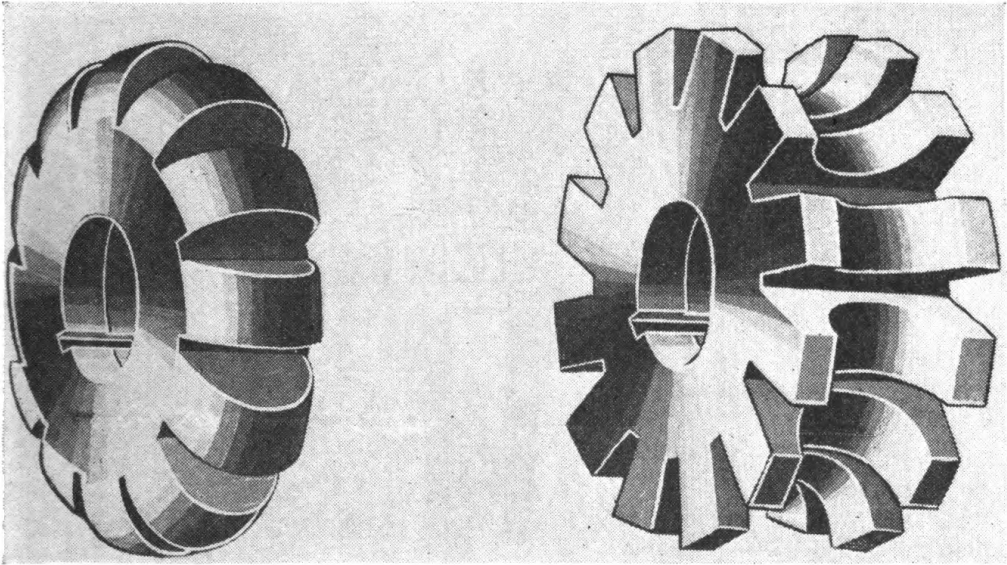
② Right hand.

FIGURE 106.—Corner rounding cutters.

31. Holding the cutter.—*a. Arbors.*—(1) The cutter is revolved by the spindle of the milling machine. Cutters are usually held by an arbor as shown in figure 108. Arbors are made in various lengths and standard diameters of $\frac{7}{8}$, 1, $1\frac{1}{4}$, and $1\frac{1}{2}$ inches with taper shanks to fit the taper hole in the spindle. An arbor may have a tang in addition to the taper which engages a slot provided in the spindle. A knock-out bar is inserted through the spindle to remove the arbor similar to the way in which live lathe centers are removed.

(2) Another type of arbor, also shown in figure 108, does not have any tang but is secured in the taper of the spindle by means of a draw-in bar which is inserted through the rear end of the spindle and screws into the small end of the arbor. This arbor also has lugs for positive drive which fit into grooves in the spindle.

The main portion of the arbor between the shank and outer end is cylindrical and collars fit over this portion. The cutter or cutters are clamped between two or more collars by a clamping nut at the outer end which presses the collars against the arbor shoulder and, consequently, the cutters. Tightening the clamping nut before the arbor support has been put in place will bend or twist the arbor and it must be straightened before being used. By using collars of different lengths, cutters of any width (thickness) can be clamped anywhere on the arbor.



① Convex.

② Concave.

FIGURE 107.—Cutters.

(3) The arbor support (fig. 94 and par. 27*h*) is secured to the overarm and supports the outer end of the arbor. This support should always be located as close to the cutter as the nature of the work allows. For light cuts with small diameter mills, the friction between the arbor collars and the cutters is sufficient to prevent the cutter from slipping on the arbor. For heavy cuts and larger diameter mills the arbor has a spline that fits a keyway in the hole of the cutter and prevents slipping.

b. Holding end mills.—End mills (fig. 102) up to 2 inches in diameter are usually made solid with their shanks. If the shank is the same size as the tapered hole in the machine spindle, such mills can be inserted directly into the spindle. If the cutter shank is not the same size as the hole in the spindle, a collet must be inserted to hold the cutter. Milling machine collets have an external taper which fits the spindle and an internal taper into which

the cutter shank fits. They serve the purpose of stepping down taper sizes and are available either with tangs or a tapped hole to engage a draw-in bar.

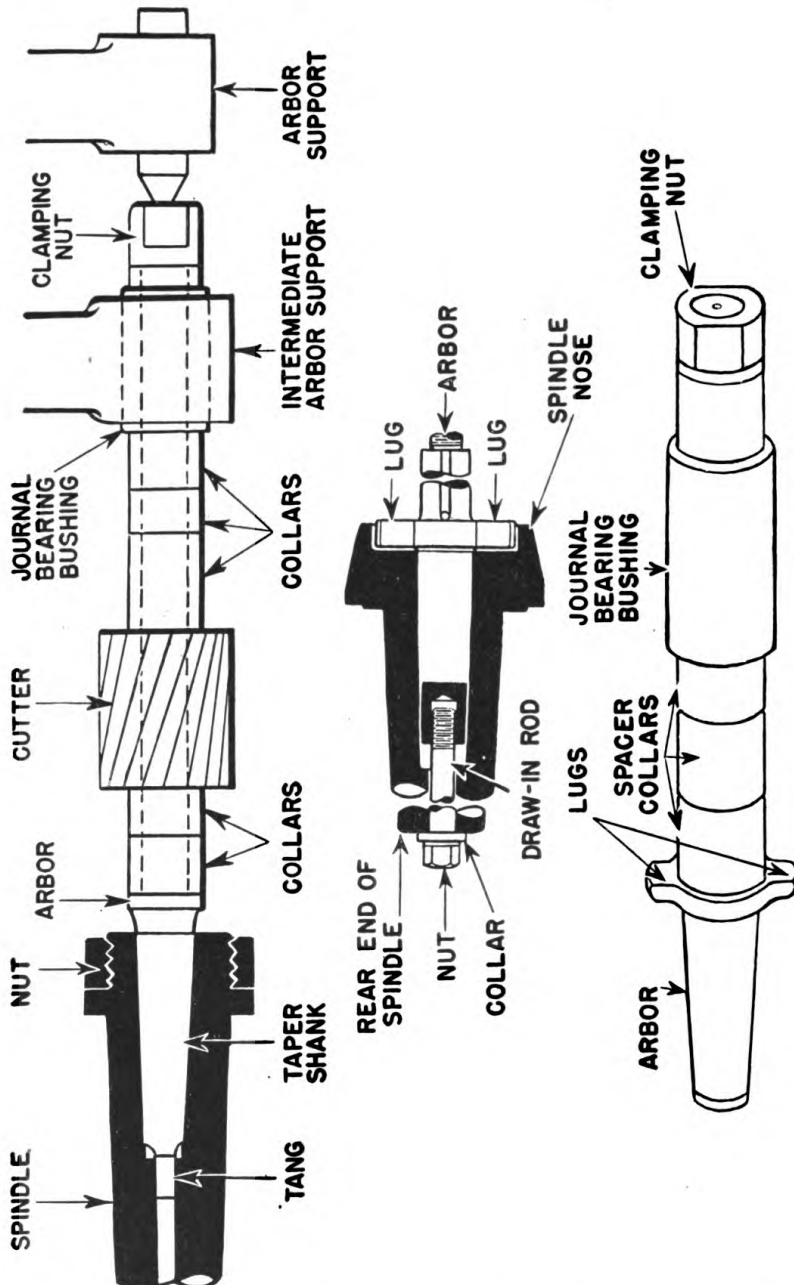


FIGURE 108.—Milling machine arbor in position for work.

c. Holding straight shank cutters.—Milling cutters with straight shanks are usually held by a spring collet. The outer end of a special arbor is tapered and threaded, and split by three equally spaced slots over which fits a nut similarly tapered and threaded.

When tightened, the nut causes the split section to contract and hold the cutter shank firmly.

d. Holding facing cutters.—Figures 109 and 110 illustrate the two methods in common use of holding facing cutters (par. 30e) directly to the machine spindle. In machines having spindles of the type shown in figure 109 (Cincinnati type), the end of the spindle is shouldered and fitted with hardened keys arranged radially. The back of the cutter is counterbored to fit over the shoulder on the end

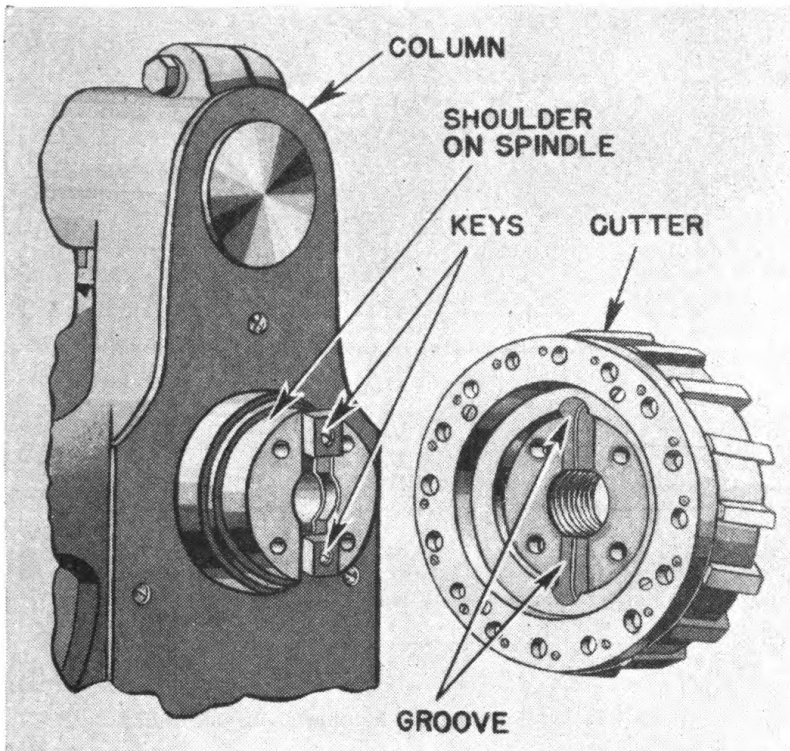


FIGURE 109.—Method of holding face mills.

of the spindle. These counterbores center the cutter and four cap-screws hold it in place. A groove in the back of the cutter engages the keys on the spindle to drive the cutter. On machines of the type shown in figure 110 (Brown and Sharpe type), the spindle nose is tapered, and the cutter is drawn over it by means of a cutter driver and draw-in bar. The cutter driver fits both a slot in the cutter and a recess in the spindle nose to drive the tool. The taper of the spindle nose serves to center the cutter.

e. Holding shell end mills.—Figure 111 illustrates an arbor for holding shell end mills (par. 30d and fig. 103). The cutter is held by the capscrow and has a slot which fits over the tongue on the arbor shoulder.

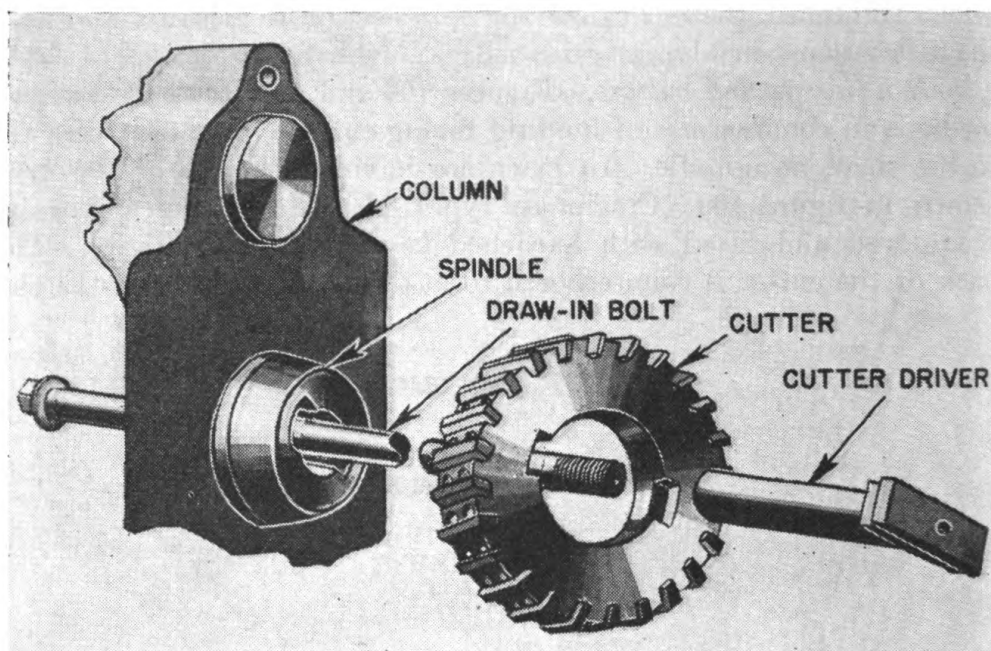


FIGURE 110.—Alternative method of holding face mills.

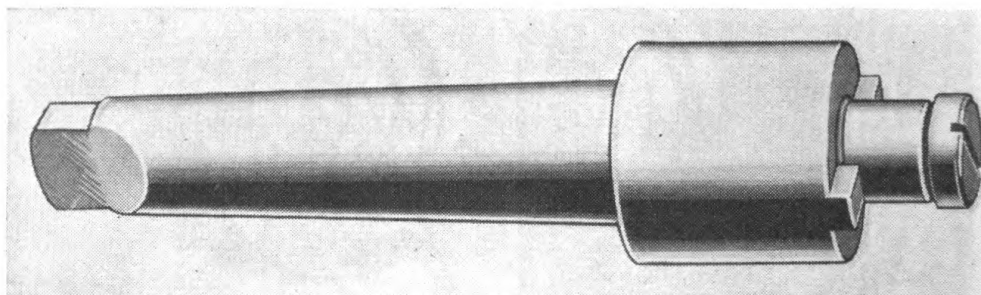


FIGURE 111.—Arbor for holding shell end mills.

f. Screw-on cutters.—Many small cutters such as corner rounding, concave, and convex cutters are provided with threaded holes and are screwed on the type of arbor shown in figure 112. Right-hand cutters have right-hand threads and left-hand cutters have left-hand threads so that the resistance of the work to the cut tends to tighten the tool on the arbor.

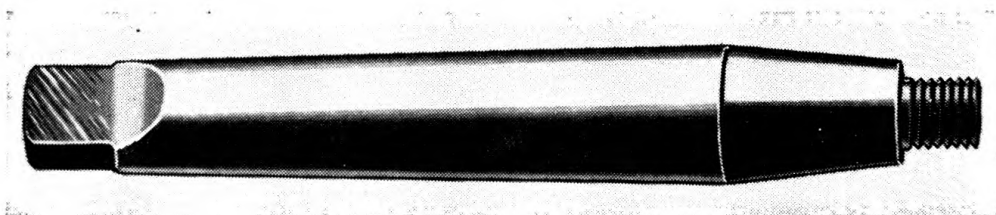


FIGURE 112.—Screw-on arbor.

32. Grinding milling cutters.—*a. Formed cutters.*—(1) Formed milling cutters are sharpened by grinding the faces of the teeth radially, every tooth alike. If one tooth is longer than the others it does more cutting and consequently becomes dull quickly. It is poor shop practice to allow any milling cutter to become very dull; the cutters should be kept sharp at all times.

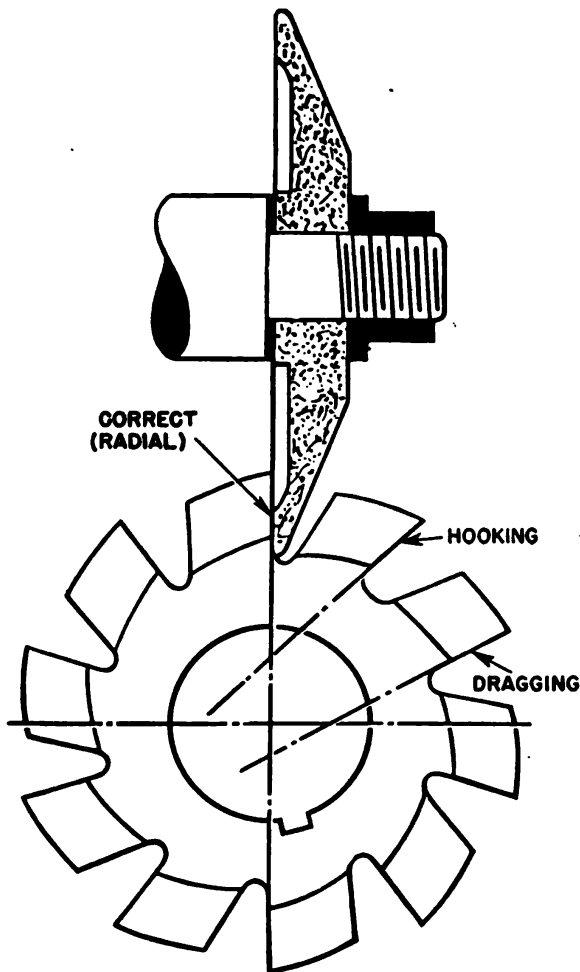


FIGURE 113.—Method of sharpening formed cutters.

(2) Figure 113 shows the generally used set-up for sharpening a formed milling cutter. Provision must be made for holding the cutter firmly in position; it is not possible to hold cutters by hand when grinding them. Some machinists set up a grinding attachment on a lathe to do this work. The fundamental requirement for efficiently sharpening formed cutters is to grind the faces of the teeth exactly in line with a radius. Otherwise the teeth will be

"dragging" or "hooking" as shown in figure 112, which alters the profile of the tool.

b. Inserted and saw tooth cutters.—Cutters with saw teeth or inserted teeth are sharpened by grinding the lands, that is, the tops of the teeth (fig. 96). The important consideration in such grinding is the angle of clearance which is the amount in degrees that the top of each tooth is ground off or relieved. If the clearance is too great, the cutter dulls rapidly; if too small, the cutter rubs rather than cuts. Proper clearance angles are about 7° for cutters under 3 inches in diameter, 5° for those over 3 inches in diameter. The clearance angle of end and side teeth should be about 2° and the face of the cutter ground 0.001 or 0.002 inch concave toward its center to avoid any drag. To obtain this clearance, grind the teeth against a point on the grinding wheel below the wheel's axis as shown in figure 114. To obtain the clearance when grinding with

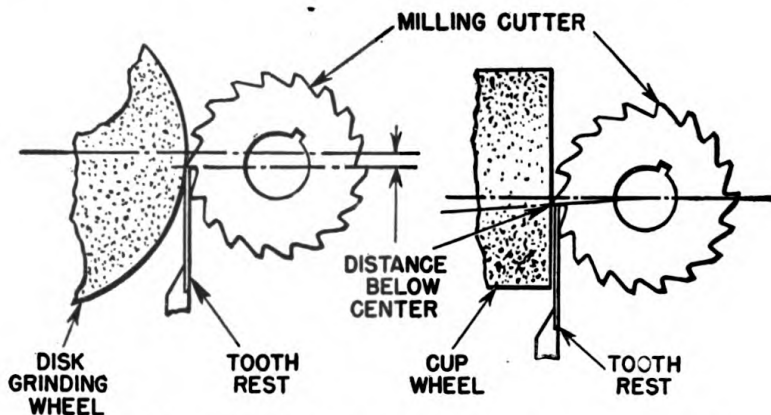


FIGURE 114.—Method of grinding clearance on milling cutters.

a disk wheel, lower the tooth rest or raise the wheel a distance equivalent to 0.0088 times the clearance angle times the diameter of the grinding wheel. For example, to find the distance the tooth rest should be below center to grind a cutter with 5° clearance on a 6-inch wheel, the calculation is as follows: $0.0088 \times 5 \times 6 = 0.264$ inch, which is the distance the tooth rest should be below center. When grinding with a cup wheel, the calculation is 0.0088 times the clearance angle times the diameter of the cutter. When grinding any cutters, a soft, medium grain wheel is best; light, fast cuts will prevent overheating the cutter.

c. Care of milling cutters.—Improper use or lack of care will soon damage any milling cutters. It is important that they be kept sharp at all times; and it is equally essential that each cutting edge be exactly the same distance from the axis of the cutter. Take care

that the arbors run true and that the portion on which the cutter or collars rest is smooth, clean, and free from nicks or burs.

33. Holding the work.—*a. General.*—As explained in paragraph 27e, milling machine tables are provided with several T-slots of standard width, running exactly parallel to the direction of longitudinal table travel, for the purpose of clamping the work itself or any device for holding the work. One of the following methods is generally used to locate and hold the work to be milled:

- (1) It may be clamped directly to the table.
- (2) It may be held in a vise.
- (3) It may be fastened to an angle plate or similar device which is secured to the table.
- (4) It may be held in a special fixture.

b. Clamping.—(1) Practically all metals used in machine-shop practice, iron, steel, brass, aluminum, etc., will spring more or less under pressure. If a piece is clamped in such a way that it will spring or buckle under cut, the machined surface will be inaccurate when the clamps are removed. For this reason remember, in clamping work to the table of any machine, miller, shaper, or planer, that the work must seat perfectly on the table, or else be shimmed or blocked, especially under the clamps. Because of the unlimited variety of shapes and sizes of milling machine work, it is impossible in this manual to explain the exact method of clamping each piece, but certain principles apply to all jobs.

(2) Figure 115 shows types of clamps commonly used for securing work to machine tables. The flat, plain, or strap clamp is provided with an elongated bolt hole somewhat nearer the front or work end; this end is usually beveled as shown. The U-clamp may be removed from the clamp bolt without removing the nut and washer; the bent or goose neck clamp is useful when it is desirable to have the nut on the clamping bolt below the travel of the tool. The finger or pin clamp is used where it is not possible to clamp the work in the regular way and permit the top of the clamp to clear the cutter. In such cases, it may be possible to drill holes in each end of the work and hold it by inserting the turned portion of the finger clamp into the holes. The clamps are held by bolts placed through the holes; the heads of these bolts fit the table T-slots, and the clamp is forced down on some portion of the work by a nut and washer. (It is poor practice ever to clamp work without using a washer under each nut.)

(3) Three forms of clamp bolts in general use are shown in figure 116. The squarehead bolt must be pushed into position from an end of the T-slot. The tapped squarehead bolt used in conjunction with

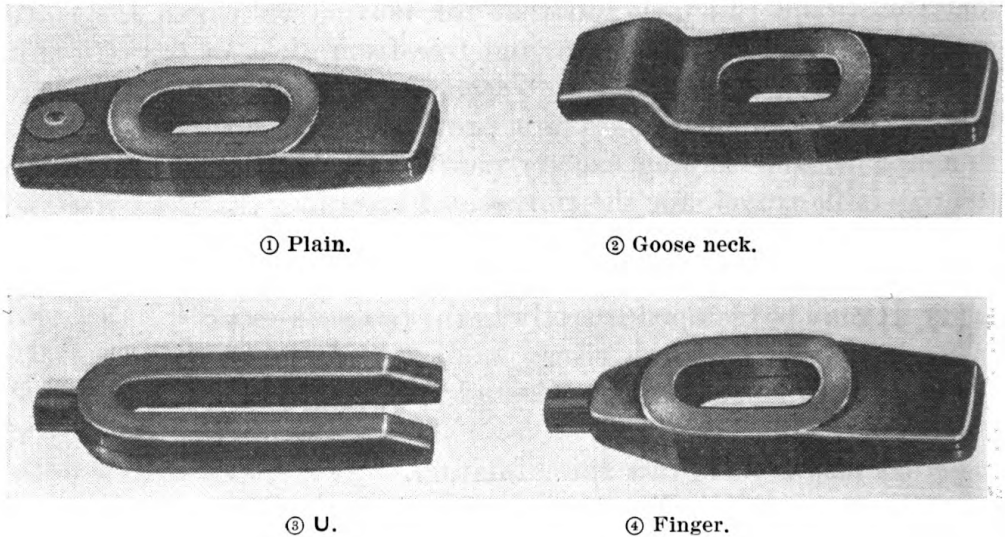


FIGURE 115.—Common forms of clamps.

a stud as shown, is exceptionally strong. The T-head bolt can be placed in the table T-slot at any position.

(4) To seat the work solidly, use shims or blocks. Shims are thin pieces of metal, wood, or paper placed between the table and the

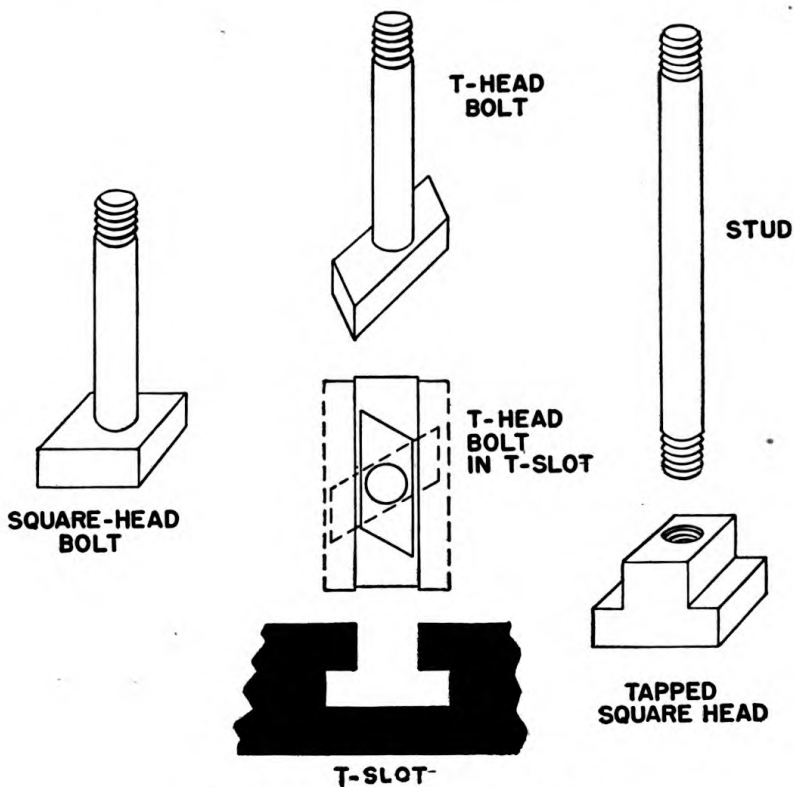


FIGURE 116.—Forms of clamp bolts.

work, or between any two pieces or parts for purposes of adjustment or to give support. Heavy shims are generally called packing blocks or simply blocks. Clamps should always be properly placed and the blocks used under them must be the correct height or the work may become loose. The clamp bolt should be as close to the work as con-

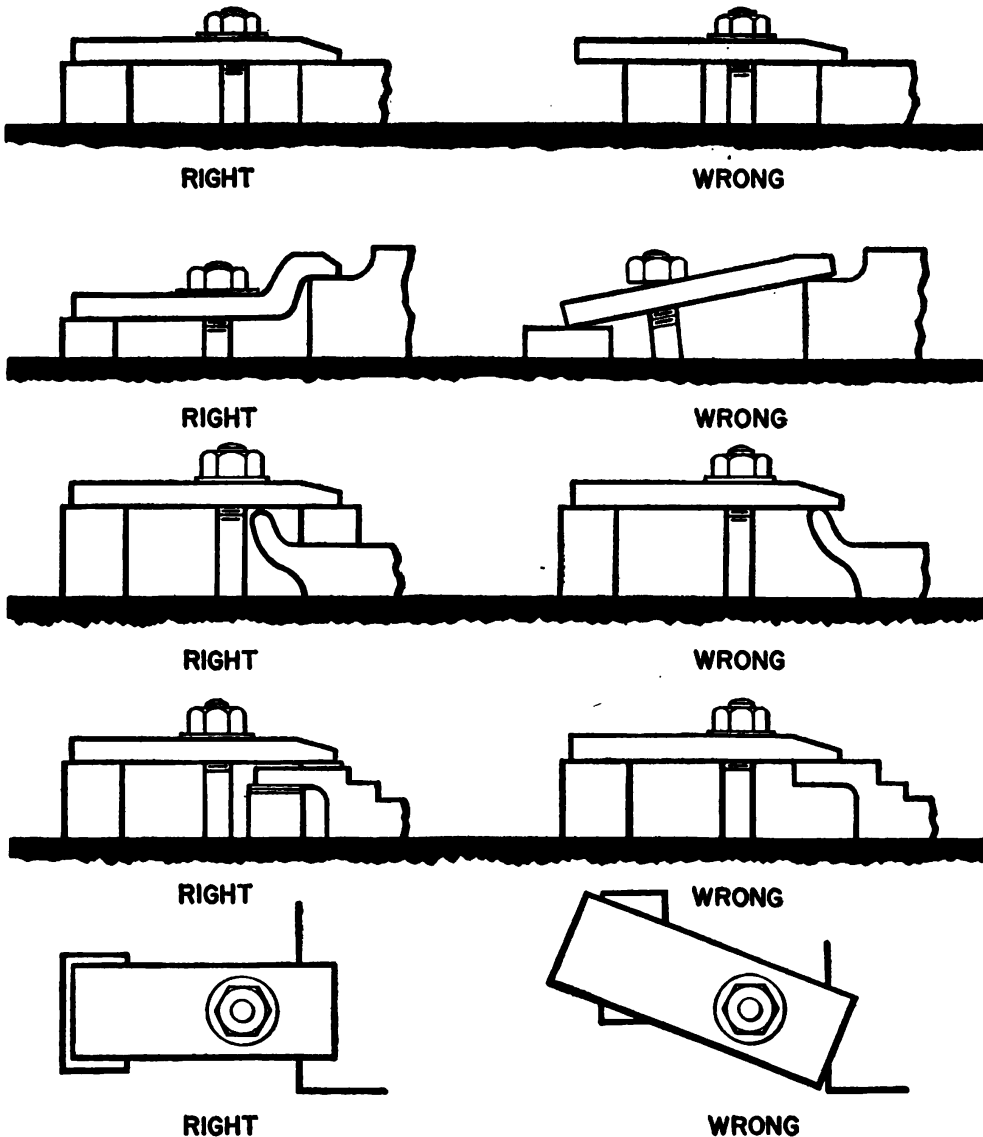


FIGURE 117.—Right and wrong applications of clamps.

ditions permit, and the clamp should have a firm seat on both the work and the block. As a rule, the block should never be lower than the surface of the work on which the clamp rests, nor should a clamp be turned down on any portion of the work which might spring unless a block is also placed under it. Figure 117 illustrates some correct

methods of clamping as well as some common errors which should be avoided.

c. Clamping hints.—The following suggestions have proved valuable in obtaining efficient set-ups on machine tables:

(1) Clamps should not be tightened on an unprotected finished surface.

(2) Be sure to protect in some way the table or vise from scores when clamping rough castings or forgings; paper is generally used.

(3) Before clamping any work, be sure that it, as well as the machine table and all clamps, bolts, shims, and blocks are thoroughly clean.

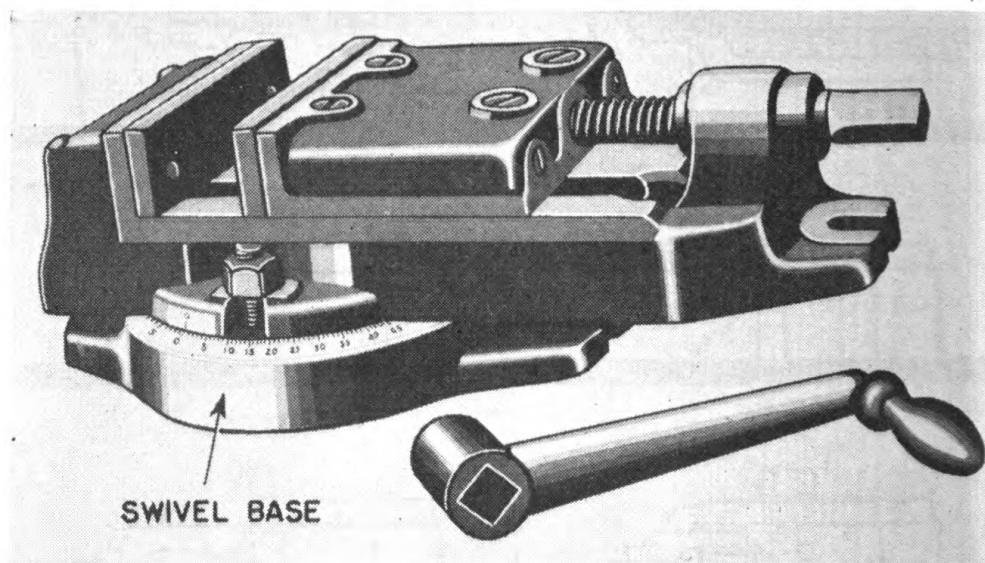


FIGURE 118.—Swivel vise.

(4) Avoid unnecessary clamping.

(5) Always be sure the clamp bolts are long enough fully to engage the threads of the nuts without projecting too high.

(6) Oil the threads of clamp bolts occasionally.

(7) Be sure that all clamps, bolts, and nuts clear the path of the cutting tool before starting the machine.

d. Vises.—(1) Vises may be used for holding a large variety of milling machine work, especially if it is comparatively small. The two types of vises most generally used are the plain vise and the swivel vise shown in figure 118.

(2) The plain vise is the same as the swivel vise except that it is bolted directly to the milling machine table instead of to the swivel base. It is best for milling cuts parallel to or at right angles to the length of the work. It is provided with lugs by which it is bolted to

the table, and by means of removable keys which engage the T-slots and slots cut in the base of the vise; it can be quickly set exactly lengthwise or crosswise of the table. The swivel vise has a swivel base graduated in degrees and can be set and secured at any angle to the direction of the table travel. A third type of vise (not illustrated) is the toolmaker's universal vise which can be set at any angle in either a horizontal or vertical plane. It is used mostly in toolmaking and model work.

e. Angle plates.—Angle plates are available for milling machine tables and are used in much the same way as when fastened to a lathe faceplate (par. 22a). They are especially satisfactory for holding work for face milling or as a stop for work which must be set up lengthwise of the table.

f. Fixtures.—A milling fixture is a special device for accurately locating and securely holding one or more pieces while the desired cut is being made. Fixtures are primarily used for producing quantities of duplicate pieces; they vary in form all the way from special parallels, blocks, or vise jaws to complicated mechanical devices and are usually made specially for a particular piece of work.

34. Speeds, feeds, and coolants.—*a. Speeds.*—(1) The speed of a milling cutter is the distance in feet per minute each tooth travels as it cuts its chip. The number of spindle rpm necessary to give a desired speed depends on the size of the cutter, and the best speed is determined by the kind of material being cut and the material from which the cutter is made. Since these factors are variable, it is not possible to specify exact speeds for milling operations; table XVIII gives approximate speeds for various materials when using high-speed steel cutters. Carbon-steel cutters should be run at about half the speeds recommended in the table.

(2) The method of changing feet per minute into spindle rpm is as follows: Divide $\frac{1}{4}$ of the diameter of the cutter into the cutting speed desired. For example, an operator wants to machine a piece of steel at a speed of 35 fpm with a cutter 2 inches in diameter; $\frac{1}{4}$ of 2 inches = $\frac{1}{2}$ inch; 35 divided by $\frac{1}{2}$ = 70. So he would set the machine spindle for a speed of 70 rpm.

(3) There are no hard and fast rules governing the speed of milling cutters; experience has shown that the following factors must be considered in regulating speed:

(a) A slitting cutter can be rotated faster than one with a broad face.

(b) Angle cutters must be run at slower speeds than straight cutters to prevent damage to their more slender teeth.

(4) For roughing cuts, a moderate speed and coarse feed often give best results; for finishing cuts, the best practice is to reverse these conditions, using a higher speed and light feed.

(5) Cutters with inserted blades generally will not stand as much speed as solid cutters. The highest speed possible under the circumstances is most efficient, but experimenting to determine the maximum speeds that tools will stand is unprofitable for anything but production work.

b. Feeds.—(1) In milling machines having individual motor drive, the feed mechanism is usually independent of the number of rpm of the spindle; in such machines, the feed is defined as the distance the table moves in inches per minute. Most machinists speak of feed as "1 inch" or "6 inches," meaning 1 inch or 6 inches per minute table travel. A popular milling machine is provided with a feed range from $\frac{1}{2}$ inch to 40 inches per minute. As with speeds, no hard and fast rules for correct feeds can be given; the depth, width, and type of cut, the diameter of the cutter, the way it is held, the speed, and the power and rigidity of the machine all combine to influence the best rate of feed. The general tendency is to overspeed and underfeed in milling operations; too much speed with insufficient feed will make the cutter dull very quickly. If the work requires both a roughing and a finishing cut, it is general practice to take the roughing cut with all the speed and feed that the cutter and the machine will stand and reduce the feed for the finishing cut. At least $\frac{1}{64}$ inch of metal should be left for the finishing cut.

(2) Feed for milling cutters is from 0.002 to 0.250 inch per cutter revolution and depends on the diameter of the cutter, the kind of material, the width and depth of cut, the size of the work, and whether a light or heavy machine is used. Plain mills will stand a coarse feed; angular mills will not. A 3-inch plain cutter at 40 fpm, feed 0.040 inch per cutter revolution, will produce a finished surface.

c. Direction of feed.—(1) It is usually regarded as standard practice to feed the work against the milling cutter as shown in figure 119. When the work is fed in this manner, the teeth cut under any scale on the work, and any backlash in the feed screw is taken up by the force of the cut.

(2) As an exception to this recommendation, it is advisable to reverse the direction of feed shown in figure 119 when cutting off stock or when milling comparatively deep or long slots.

(3) The direction of cutter rotation is related to the manner in which the work is held. The cutter should rotate so that the work

springs away from the cutter then there will be no tendency for the force of the cut to loosen the work. No milling cutter should ever be rotated backward; this will break the teeth. If it is necessary to stop the work during a finishing cut, the power feed should never be thrown out, nor should work ever be fed back under the cutter unless the cutter is stopped or the work lowered. Never change feeds while the cutter is rotating.

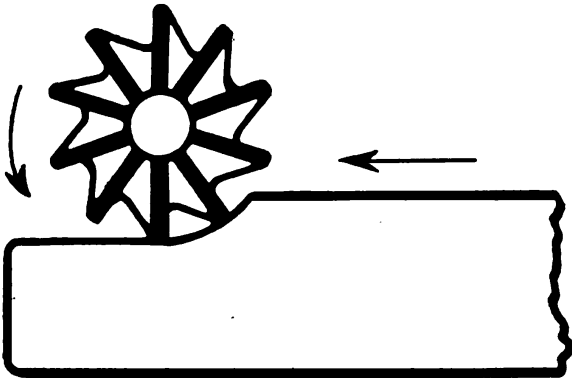


FIGURE 119.—Usual direction of feed in milling.

d. Coolants.—Always use a lubricant or coolant when milling steel or wrought metals. (Cast iron is always milled dry.) This carries away the troublesome chips, produces a better finish, and keeps the cutter cool, which increases its capacity for work between sharpenings. Some milling machines are equipped with pumps which provide a steady stream of coolant over the cutter; others without pumps can be arranged so that a can allows the coolant to flow by gravity through a pipe to the cutter. All milling machines are designed to save the coolant so it can be used again, usually by means of a drain in the base (fig. 94). Table XVIII specifies recommended coolants for milling various metals.

35. Milling operations.—*a. Plane or surface milling.*—Machining flat horizontal surfaces with plain milling cutters is called plane or surface milling. When the cut is considerably wider than the diameter of the cutting tool, the operation is called slabbing. An exceptionally wide cutter or a combination of two or more interlocking cutters of the same diameter is called a slabbing cutter. Figure 120 shows a plane or surface milling operation.

b. Straddle milling.—When two or more parallel vertical surfaces are machined at a single cut, the operation is known as straddle milling. This is done by mounting two or more side mills some distance apart on an arbor so that they straddle the work. Figure 121 shows the

set-up and principle; in this illustration the machinist is milling two sides of the work and cutting a slot at one set-up.

c. Angle milling.—Angle milling is machining a flat surface at some angle other than 90° to the axis of the milling cutter. This

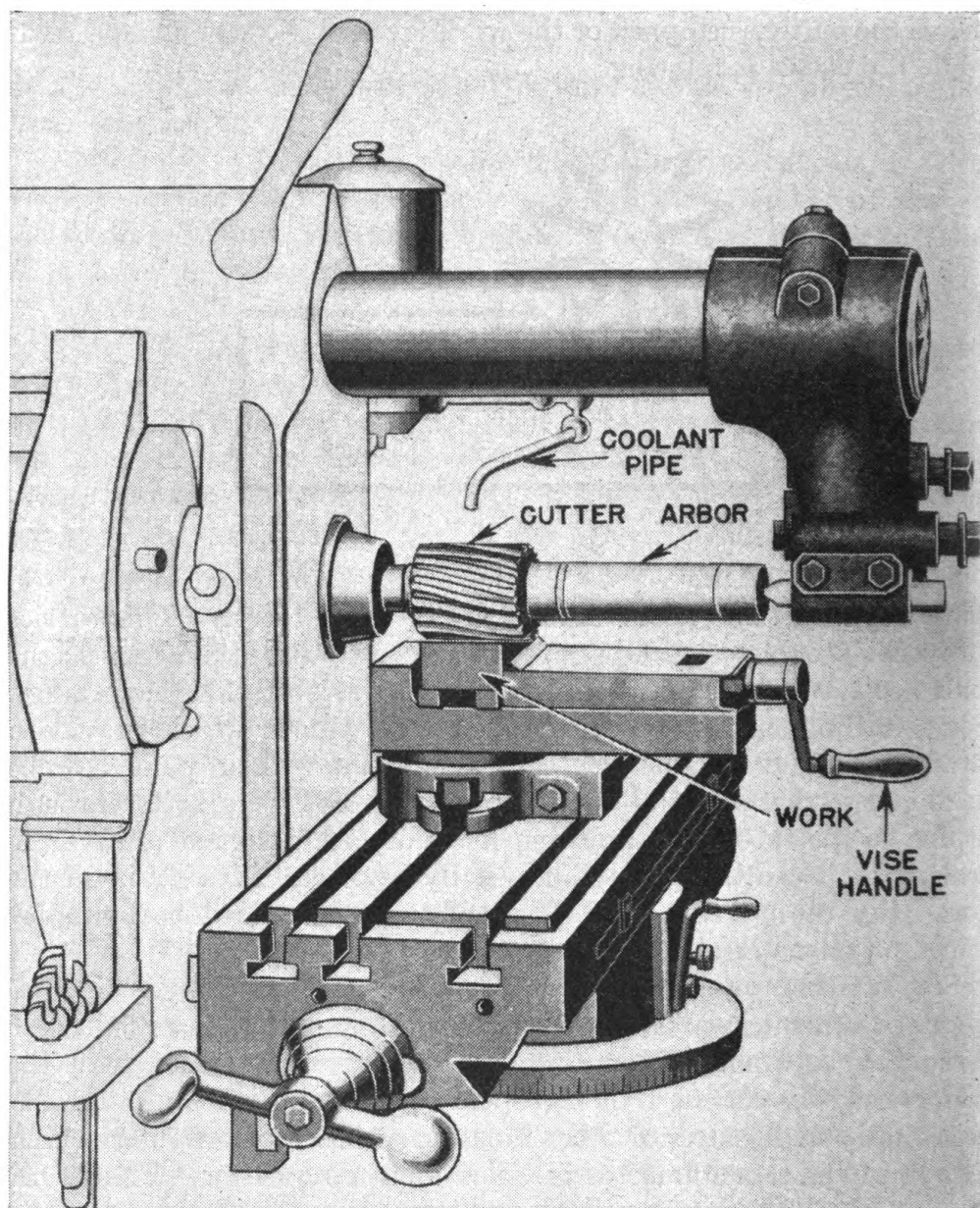


FIGURE 120.—Surface milling.

is done by using some form of angular cutter and setting up the work so the surfaces are properly located. Figure 122 shows the set-up generally used for milling a dovetail with a special angular cutter called a dovetail cutter.

d. Side or face milling.—Machining vertical surfaces at right angles to the axis of the cutter is called side or face milling. Face mills are generally used. Figure 123 shows a common facing operation being done with an inserted tooth face mill.

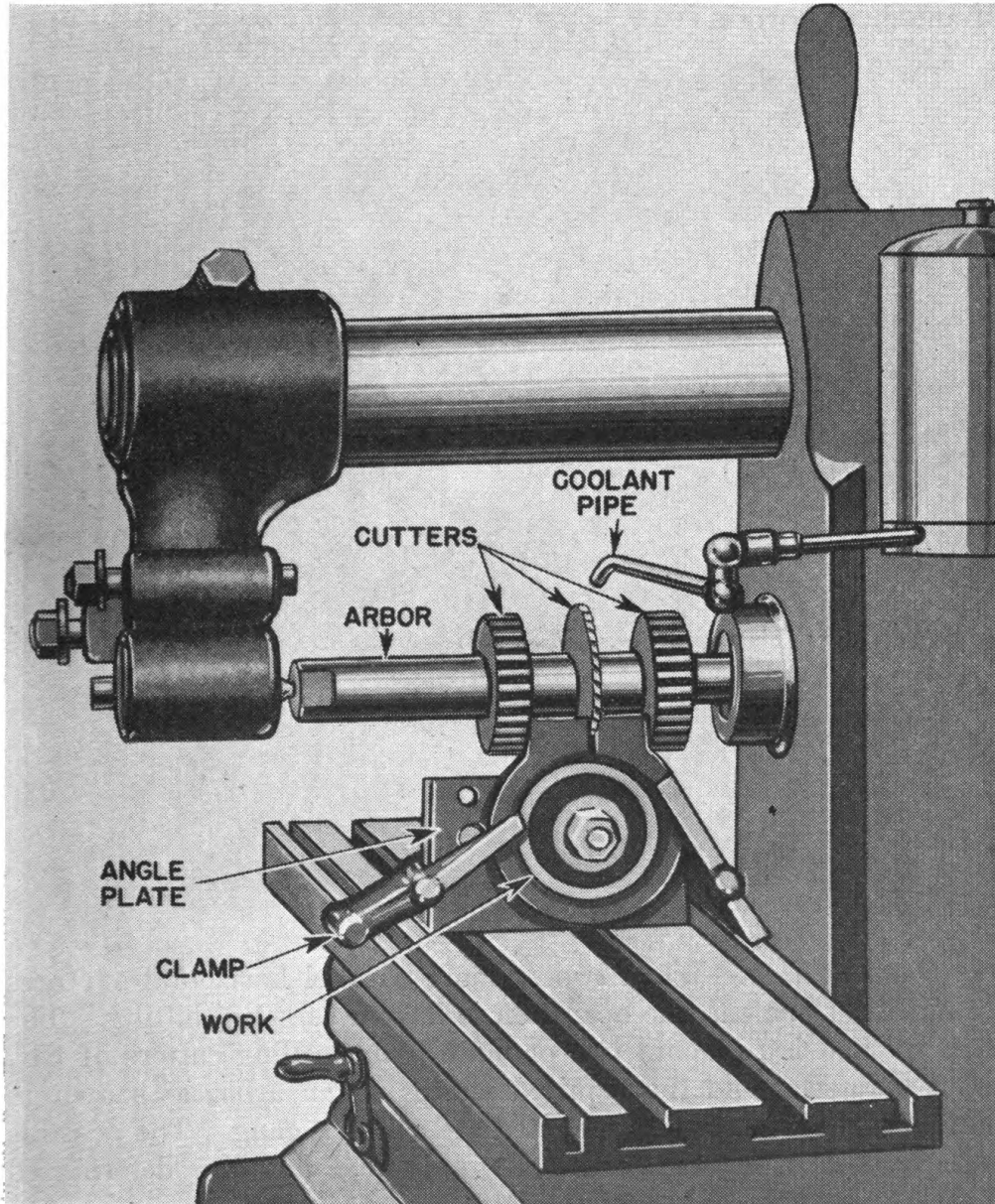


FIGURE 121.—Straddle milling.

e. Form milling.—This is the process of machining special contours composed of curves and straight lines, or entirely of curves, at a single cut. This is done by specially formed cutters as shown in figure 124.

f. Profiling.—This is machining the vertical edges of work of irregular contour and is generally done with an end mill held in a vertical spindle. The exact form of the work is determined by a template or profile attached to the work or to the fixture supporting it. Profiling is a special operation seldom necessary in the general run of automotive work.

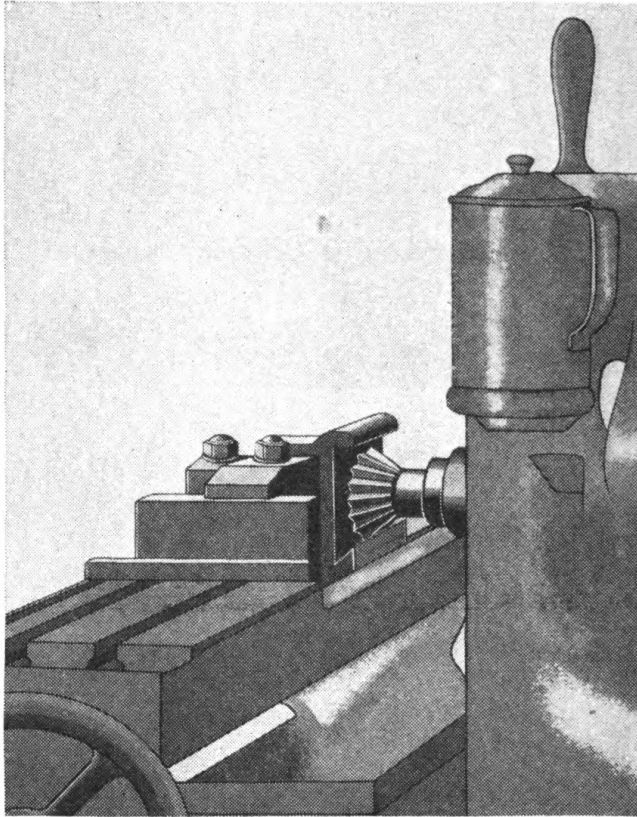


FIGURE 122.—Angle milling.

g. Gang milling.—When two or more parallel horizontal surfaces are milled at one cut, the operation is known as gang milling. The usual method is to mount two or more plain milling cutters of different diameters and the required widths on an arbor as shown in figure 125. The cutters so mounted are called a gang. The possible cutter combinations are practically unlimited and are determined in each case by the nature of the job.

h. Keyway cutting.—Cutting keyways and splines in shafts is a common operation in motor vehicle repair shops and the milling machine is ideal for the purpose. Several cutters can be used, depending on the type of keyway desired—a plain mill, an end mill, a cotter mill, or a Woodruff keyway cutter. There are also several

methods of setting up the work—in a vise, in V-blocks, clamped to the table, or held between index centers. In any case, the center of the shaft must be exactly under the center of the face of the cutter, and the keyway must be milled parallel to the axis of the shaft.

36. Indexing.—*a. Principles and range of operations.*—(1) Indexing is the process of rotating a piece of work on its axis through

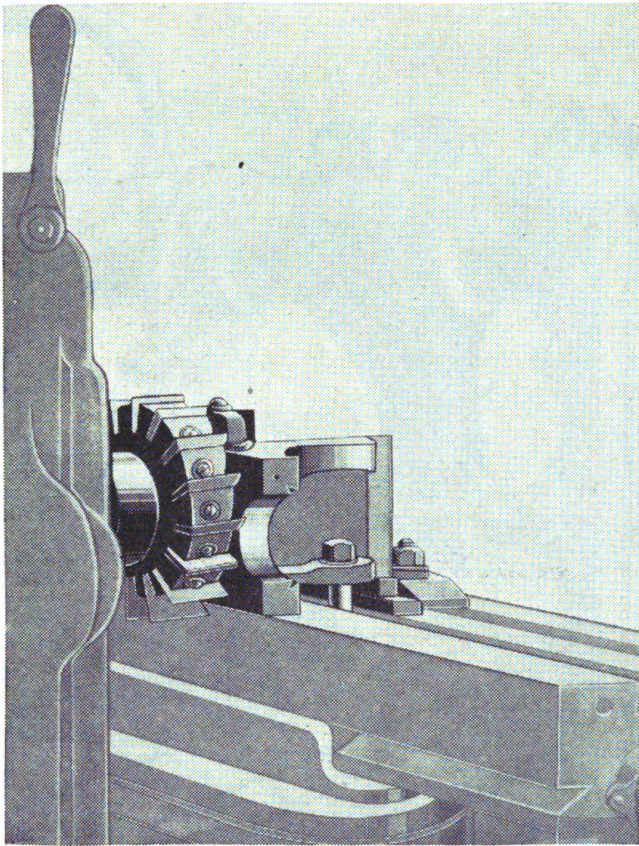


FIGURE 123.—Face milling.

some exact fraction of a whole revolution. For example, the operator may want to rotate a gear blank the distance from one tooth space to another when cutting the teeth, or rotate the work from one groove to the next when fluting a reamer. The desired rotation is obtained by a dividing head. The dividing head and its footstock are known as index centers. Figure 126 illustrates a typical dividing head and footstock and the nomenclature of their essential parts.

(2) The dividing head contains a worm and worm wheel for transferring motion from the index crank to the spindle. The work spindle is taper-bored to hold a live center or the taper shank of a tool and has a threaded nose to hold a chuck or other work-holding

device. It is carried on large bearing surfaces in the swivel block which is constructed so that it will hold the spindle at any angle from 5° below the horizontal to at least 10° beyond the perpendicular. Graduations in half degrees measure these angular positions. The index plate, index crank, and sector permit work to be rotated

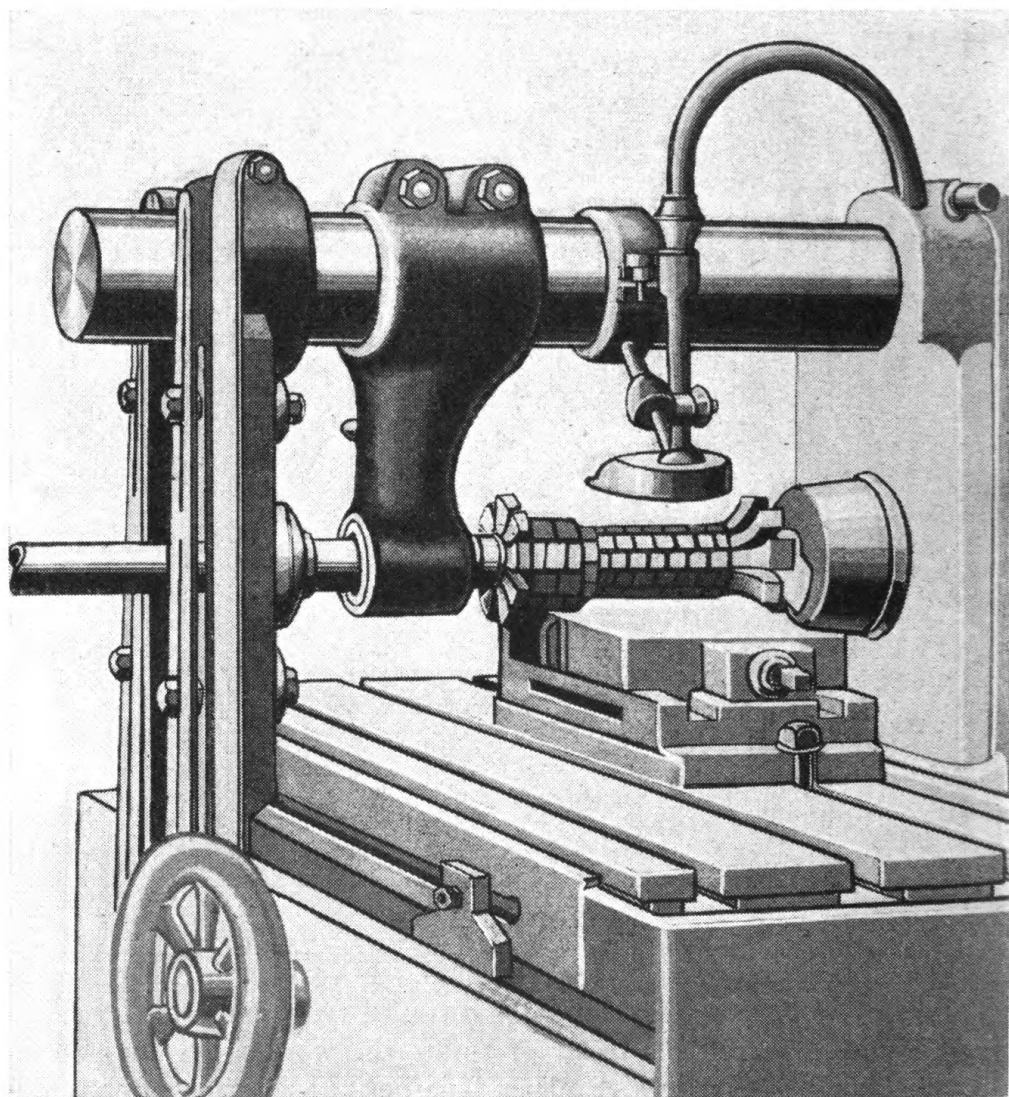


FIGURE 124.—Form milling.

through any part of a revolution. The footstock supports the outer end of a piece being milled. It can be adjusted toward or away from the live center, set over horizontally as a lathe tailstock, and adjusted in a vertical plane for the purpose of milling tapers. Indexing is commonly used with the milling machine for gear cutting,

splining shafts, milling bolt heads, fluting taps and reamers, or generally for operation where it is necessary to divide the circumference of the work into any number of equal parts.

b. Simple indexing.—(1) Figure 127 illustrates a simple indexing mechanism. It consists of a 40-tooth worm wheel fastened to the work spindle, a single-cut worm, a crank for turning the worm shaft, and an index plate and sector. Since there are 40 teeth in the worm

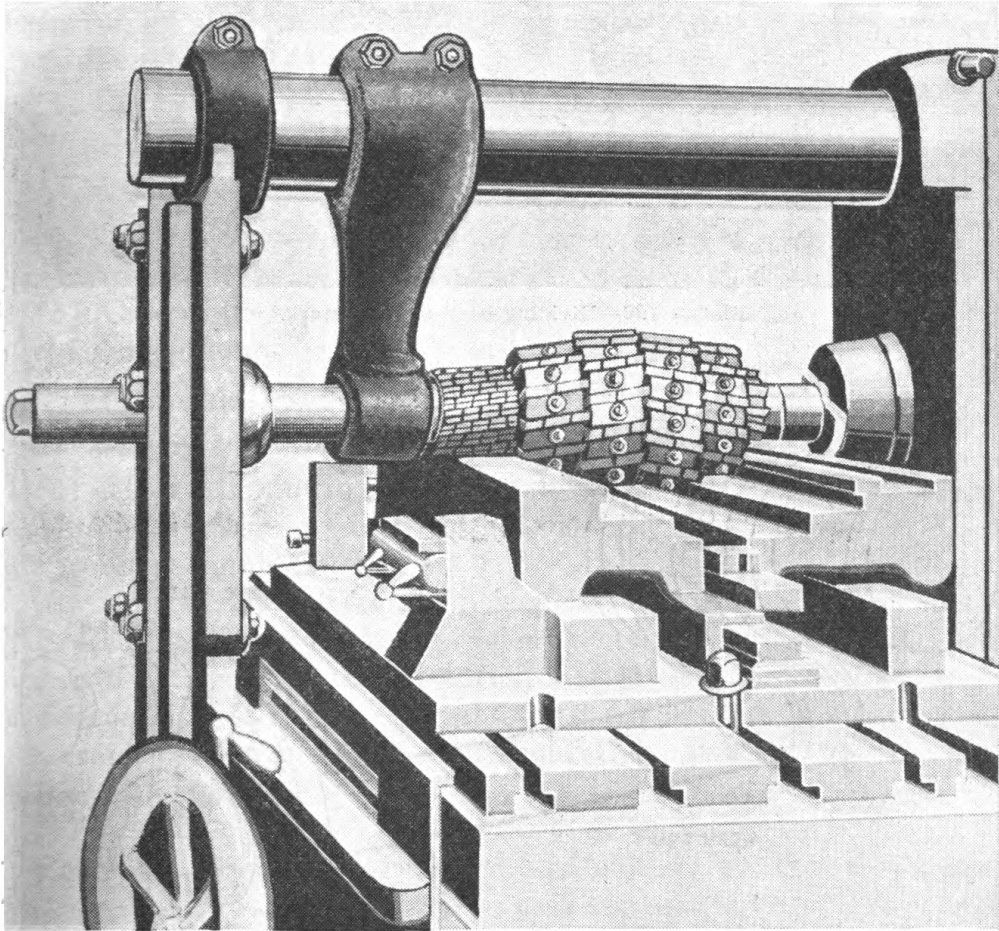


FIGURE 125.—Gang milling.

wheel, one turn of the index crank causes the worm wheel and consequently the spindle to make $\frac{1}{40}$ of a turn; so 40 turns of the index crank revolve the spindle one full turn.

(a) Suppose it is desired to cut a reamer with 8 equally spaced teeth. Since 40 turns of the index crank turn the spindle one full turn, $\frac{1}{8}$ of 40 or 5 turns of the crank after each cut will space the reamer for 8 teeth. If it is desired to space equally for, say, 10 teeth, $\frac{1}{10}$ of 40 or 4 turns would produce the correct spacing.

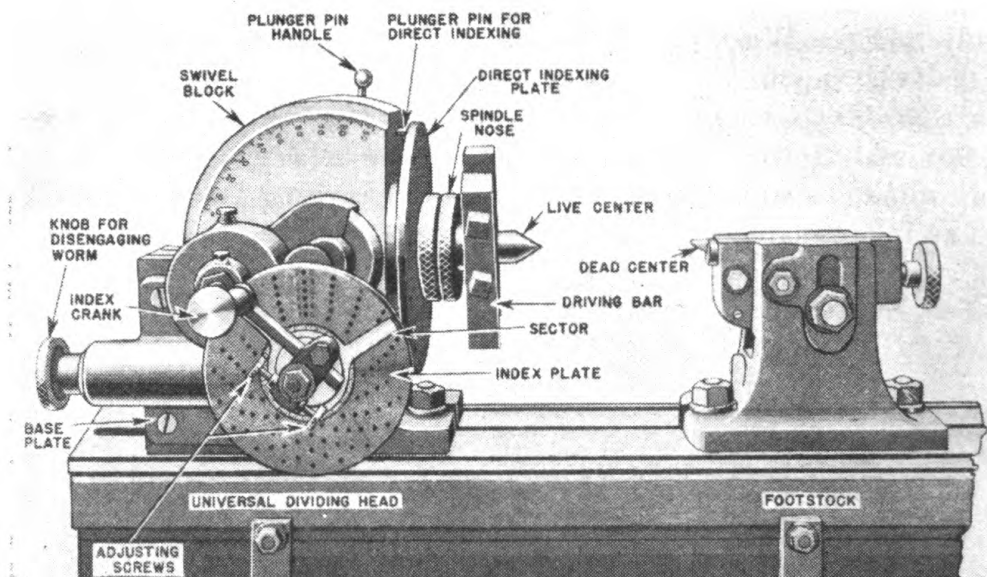


FIGURE 126—Dividing head and footstock.

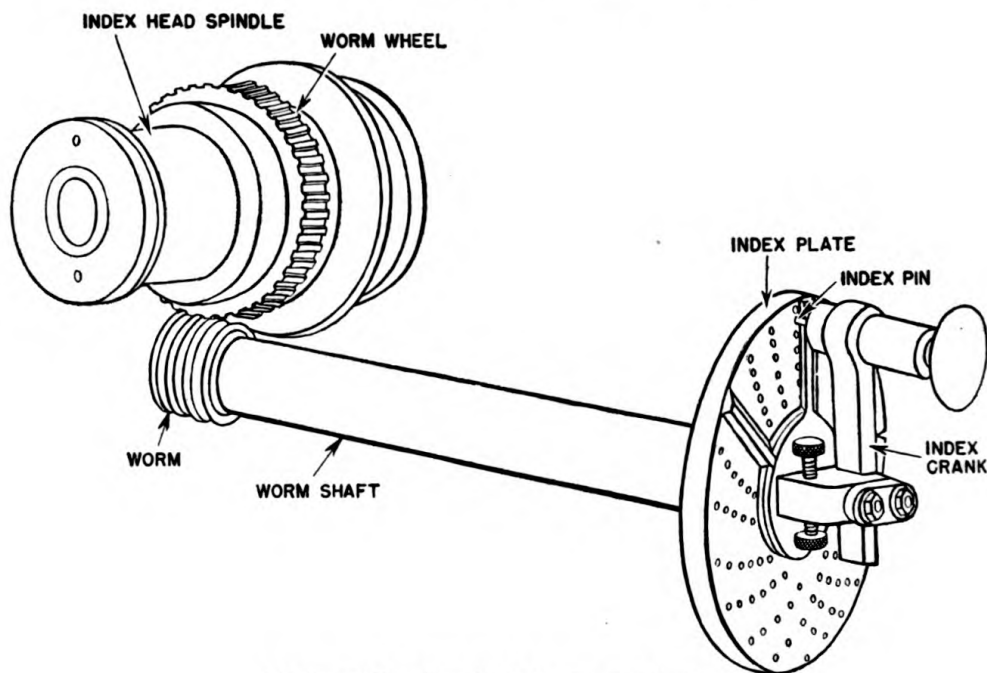


FIGURE 127.—Simple indexing mechanism.

(b) The same principle applies whether the divisions required divide evenly into 40. For example, if desired to index for 6 divisions, 6 divided into 40 equals $6\frac{2}{3}$ turns; similarly, to index for, say, 14 spaces, 14 divided into 40 equals $2\frac{6}{7}$ turns. These examples may be multiplied indefinitely and from them the following rule is derived:

(c) To determine the number of turns of the index crank needed to obtain one division of any number of equal divisions on the work, divide 40 by the number of equal divisions desired (provided the worm wheel has 40 teeth which is standard practice).

(2) The index plate (fig. 127) is a circular plate with a series of six or more circles of equally spaced holes; the index pin on the crank can be inserted in any hole in any circle. With interchangeable plates regularly furnished with most dividing heads, the spacings necessary for most gears, bolt heads, milling cutters, splines, etc., can be obtained. The following sets of plates are standard equipment:

(a) Brown and Sharpe type, 3 plates of 6 circles each, drilled as follows:

Plate 1—15, 16, 17, 18, 19, 20 holes.

Plate 2—21, 23, 27, 29, 31, 33 holes.

Plate 3—37, 39, 41, 43, 47, 49 holes.

(b) Cincinnati type, one plate drilled on both sides with circles divided as follows:

First side—24, 25, 28, 30, 34, 37, 38, 39, 41, 42, 43 holes.

Second side—46, 47, 49, 51, 53, 54, 57, 58, 59, 62, 66 holes.

(3) The two following examples show how the index plate is used to obtain any desired part of a whole spindle turn and, consequently, the work by simple indexing:

(a) *First example:* To mill a hexagon.

Procedure: Using the rule previously given, divide 40 by 6, which equals $6\frac{2}{3}$ turns, or six full turns plus $\frac{2}{3}$ of a turn on any circle whose number of holes is divisible by 3. Therefore, 6 full turns of the crank plus 12 spaces on an 18-hole circle, or 6 full turns plus 26 spaces on a 39-hole circle.

(b) *Second example:* To cut a gear of 42 teeth.

Procedure: Using the rule again, divide 40 by 42 which equals $\frac{40}{42}$ or $20\frac{2}{21}$ turns, or 40 spaces on a 42-hole circle, or 20 spaces on a 21-hole circle. To use the rule given, select a circle having a number of holes divisible by the denominator of the required fraction of a turn reduced to its lowest terms. The number of spaces between holes gives the desired fractional part of the whole circle. When counting holes, start with the first hole ahead of the pin.

(4) The sector (fig. 128) indicates the next hole in which the pin is to be inserted and makes it unnecessary to count holes when moving the index crank after each cut. It consists of two radial, beveled arms which can be set at any angle to each other and then moved together around the center of the index plate. Suppose that as shown in figure

128 it is desired to make a series of cuts, moving the index crank $1\frac{1}{4}$ turns after each cut. Since the circle illustrated has 20 holes, turn the crank one full turn, plus 5 spaces, after each cut. Set the sector arms to include the desired fractional part of a turn, or 5 spaces, between the *beveled* edges of its arms, as shown. If the first cut is taken with the index pin against the left-hand arm, to take the next cut, move the pin once around the circle and into the hole against the right-hand arm of the sector. Before taking the second cut, move the arms so that the left-hand arm is again against the pin; this moves the right-hand arm another five spaces ahead of the pin. Then take the second cut, repeat the operation, etc., until all the cuts have been completed. (Although not necessary, it is good practice always to index clockwise on the plate.)

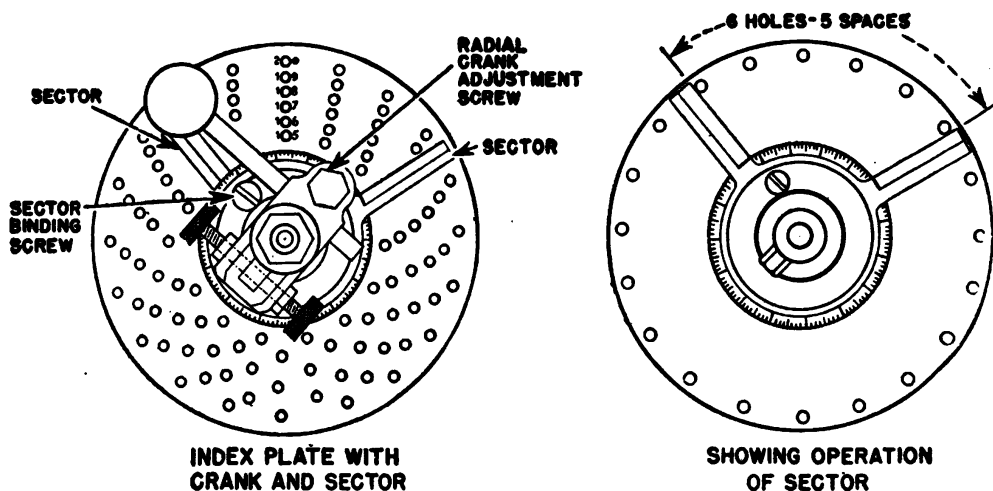


FIGURE 128.—Index plate and sector.

c. Direct indexing.—The construction of a universal dividing head permits the worm to be disengaged from the worm wheel, making possible a quicker method of indexing called direct indexing. The dividing head is provided with a knob which when turned through part of a revolution operates an eccentric and disengages the worm. Direct indexing is accomplished by an additional index plate fastened to the work spindle. A stationary plunger in the dividing head fits the holes in this plate. Move this plate by hand to index directly; the spindle and the work rotate an equal distance. Direct index plates usually have 24 holes and offer a quick means of milling squares, hexagons, taps, etc. Any number of divisions which is a factor of 24 can be indexed quickly and conveniently by the direct indexing method.

d. Differential indexing.—Sometimes a number of divisions is required which cannot be obtained by simple indexing with the index plates regularly supplied. To obtain these, a differential dividing head is used. The index crank is connected to the worm shaft by a train of gears instead of directly as in simple indexing. The selection of these gears involves calculations beyond the scope of this manual; few, if any, common automotive repair jobs demand it. For further information on the subject, consult the bibliography (app. II).

e. Indexing in degrees.—Work can be indexed in degrees as well as other fractions of a turn with the usual dividing head. There are 360° in a complete circle and one turn of the index crank revolves the spindle $\frac{1}{40}$ turn or 9° . Therefore $\frac{1}{9}$ of the crank rotates the spindle 1° . Work can then be indexed in degrees by using a circle of holes divisible by 9. For example, moving the crank 2 spaces on an 18-hole circle, or 3 spaces on a 27-hole circle, rotates the spindle 1° . Smaller crank movements further subdivide the circle: moving 1 space on an 18-hole circle turns the spindle $\frac{1}{2}^\circ$; 1 space on a 27-hole circle $\frac{1}{3}^\circ$, etc.

37. Operating and safety precautions.—*a. Operating practices.*—The success of any milling operation depends to a great extent upon the judgment in setting up the job, selecting the proper cutter, and holding the cutter by the best means under the circumstances. Some fundamental practices have been proved by experience to be necessary for good results on all jobs. Some of these practices are—

(1) Before setting up a job, be sure that the work, the table, the bore in the spindle, and the arbor or cutter shank are all clean and free from chips, nicks, or burs.

(2) Set up every job as close to the machine column as the circumstances permit.

(3) Do not select a cutter of larger diameter than is necessary.

(4) Keep cutters sharp at all times.

(5) Do not change feeds or speeds while the work is being cut.

(6) Always lower the table before backing the work under a revolving cutter.

(7) Feed the work in a direction opposite to the rotation of the cutter, except when milling long or deep slots or when cutting off stock.

(8) Never run a cutter backward.

(9) When using clamps, be sure that they are tight and that the work is held so that it will not spring or vibrate under cut.

(10) Use a recommended coolant liberally.

(11) Keep chips away from the table and around the work; brush them out of the way by any convenient means but do not do so by hand.

(12) Use good judgment and common sense in planning every job and profit by previous mistakes.

b. Safety precautions.—The milling machine is not dangerous to operate but always observe the following precautions to avoid injury, or damaging the machine:

(1) Keep hands and fingers away from rotating cutters. Never use hands to remove chips.

(2) Don't feed the table too far in any direction.

(3) Don't feed the knee too far up or down.

(4) Don't feed the saddle against the column or too far out from it.

(5) Never reverse the machine while it is running.

(6) Stop the machine before changing feeds or speeds.

(7) Be sure the cutter does not slip on the arbor.

(8) Take care that all clamps and clamp bolts are low enough to pass under the arbor and cutter.

(9) Be careful not to allow a brush, rule, or anything else to get between the cutter and the work while taking a cut.

(10) Keep the machine thoroughly oiled as directed by the manufacturer's manual.

SECTION V

SHAPER

	Paragraph
General.....	38
Parts and nomenclature.....	39
Cutting tools.....	40
Operating the shaper.....	41
Operating and safety precautions.....	42

38. General.—*a. Principle and range of operations.*—The shaper is a machine in which a reciprocating single-edged cutting tool removes metal from work by an operation basically different from that of either the lathe or the milling machine. (Planers used for large or heavy work operate on a similar principle except that the work reciprocates and the tool is fed into it.) Shapers are widely used for machining comparatively small flat or curved surfaces, cutting gear teeth, cutting internal or external keyways, slotting, splining, etc. Shaper tools move back and forth horizontally; similar machines with a vertical motion are usually called slotters.

b. Types and sizes.—(1) Two types of shapers are in general use: the traveling head type, in which the cross feed is in the ram; and the pillar or column type, in which the cross feed is in the table. This manual explains the operation of the pillar or column type, which is most often used in Army automotive repair shops.

(2) The size of a shaper is given as its maximum stroke in inches; sizes range from 6-inch bench types up to heavy duty machines having a 30- to 36-inch stroke. In most shapers, the length of the stroke

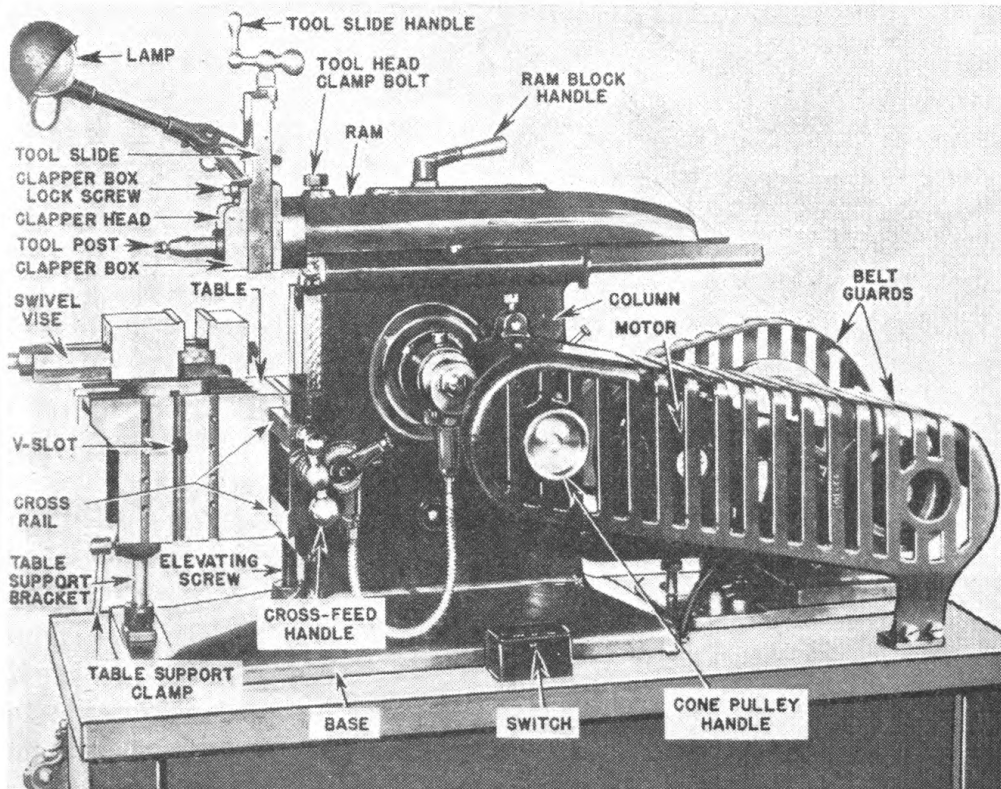


FIGURE 129.—6-inch column type bench mounted shaper.

is adjustable from zero to maximum. They are driven either from a lineshaft or by an individual motor. Practically all Army machines have individual motors.

39. Parts and nomenclature.—*a. General.*—Figure 129 illustrates a 6-inch column type bench mounted shaper, used for Army instruction purposes, with its parts and controls indicated. The machines of different manufacturers vary slightly in refinements of detail but the essential parts and controls are the same in all. Whenever possible study the manufacturer's manual for whatever machine being used.

b. Main frame assembly.—This assembly consists of the base and column and the elevating screw. The column supports the cross rail and the ram on dovetails, and contains the mechanisms which reciprocate the ram and transmit power to the automatic longitudinal table feed. The base has a bearing surface on which the lower table support bracket slides longitudinally. A handle concealed in the cabinet under the base turns the elevating screw which moves the cross rail vertically. Most shapers have no power vertical feed.

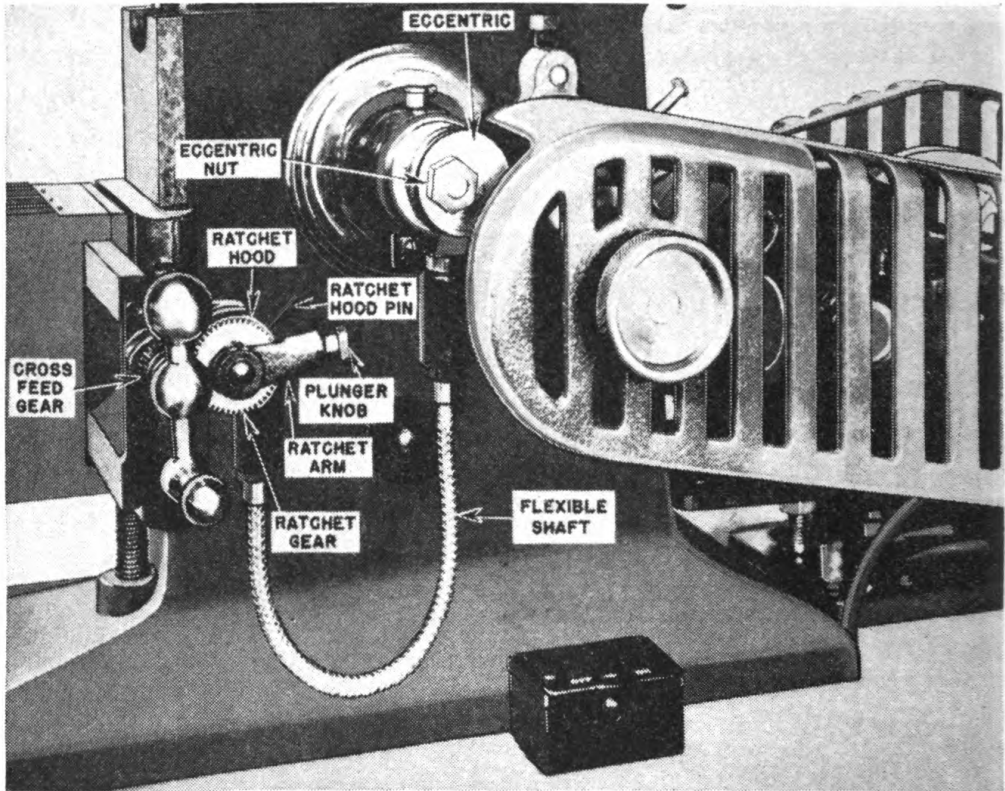


FIGURE 130.—Shaper cross feed mechanism.

c. Cross rail assembly.—This assembly includes the cross rail, the cross feed screw, the table and table support bracket, and the controls for moving the table longitudinally on the cross rail by hand or power. The cross feed gear (fig. 130) slides on the cross feed screw shaft and has an integral collar by means of which it can be meshed with the ratchet gear to engage the power table cross feed. When the cross feed gear and the ratchet gear are disengaged, the table can be moved longitudinally by turning the cross feed handle. The top of the table is provided with slots and drilled holes for clamping a vise or the work itself, and has slots with holes on both

sides, and a V-slot on one side for setting up some types of jobs. When the table has been moved to a desired vertical position, it is tightened there by the table support clamp at the top of the table support bracket. Sometimes a shaper may have a collar graduated in thousandths of an inch on the cross feed handle so that the manual cross feed can be accurately controlled.

d. Feed mechanism.—(1) This mechanism is shown in detail in figure 130. As the eccentric rotates, it transmits a reciprocating motion to the ratchet arm through a flexible shaft (a spring-steel wire in a flexible tube). A pawl is engaged with the ratchet gear by the plunger knob which moves this gear a part of a revolution in either direction at each stroke of the ratchet arm. The plunger knob controls the direction in which the ratchet gear rotates. When the cross-feed gear is meshed with the ratchet gear, the cross-feed screw rotates with it at each stroke of the ratchet arm thus feeding the table crosswise some fraction of an inch. The table is timed to move during the return stroke of the cutting tool, so that the work is in position for the cut as described in (2) below.

(2) To change the rate of feed (the distance the table moves after each stroke of the cutting tool), move the ratchet hood by means of the ratchet hood pin. The ratchet plunger will engage only those teeth on the ratchet gear which are not covered by the hood; therefore the more teeth uncovered, the greater the table feed at each stroke of the cutting tool. In the machine illustrated (figs. 129 and 130), each tooth of the ratchet wheel is equal to a cross feed of 0.003 inch. By adjusting the ratchet hood, feeds from 0.003 inch to 0.015 inch can be obtained in steps of 0.003 inch. This ratchet principle of obtaining power feed is common to all shapers; however, various manufacturers use different methods of transmitting the power.

e. Drive assembly.—This assembly consists of the ram (fig. 129) and the crank mechanism which moves it back and forth on the column dovetails. The ram is adjustable as to speed, length of stroke, and location of stroke. The cone-pulley drive from the motor provides the means of adjusting the speed in strokes per minute. To adjust the length of stroke, remove the cover plate on the side of the shaper opposite the cone pulley. The crank arm assembly, thus exposed, is shown in figure 131. Rotate the cone pulley by turning the handle until the crank shoe is at the top of its stroke in the crank arm; then loosen the eccentric nut (fig. 130) on the cone-pulley side of the shaper, which will make it possible to move the crank shoe in the crank arm by hand. Raise or lower the crank shoe until the arrow is opposite the figure on the scale on the crank arm which corresponds

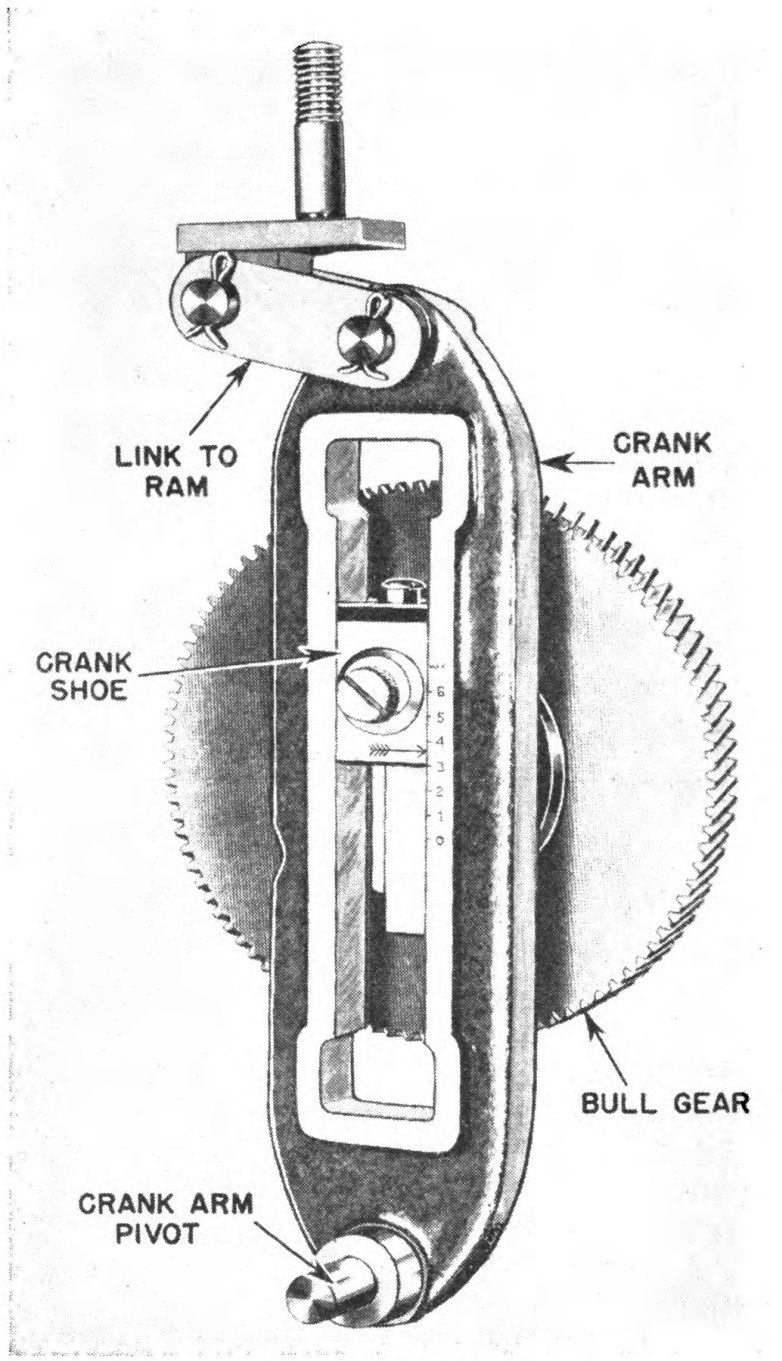


FIGURE 131.—Crank arm and shoe assembly.

to the desired stroke; then tighten the eccentric nut and replace the cover plate. If, after making this adjustment, it is found that the cutting tool will not cover the work, adjust the ram for location of stroke by loosening the ram block handle (fig. 129). The ram can now be moved in the direction of the stroke by hand to any desired position within its range on the column. This adjustment should be made with the ram at its extreme forward position. Be sure that the tool travels $\frac{1}{4}$ to $\frac{3}{8}$ inch beyond the work at each end of the stroke.

f. Tool head assembly.—(1) This assembly is on the end of the ram (fig. 129) and is the means by which the cutting tool is held and adjusted. It consists of the tool slide, the clapper box, the clapper head, and the tool post. By loosening the tool head clamp bolt on top of the ram, the entire assembly can be moved and secured at any angle, usually up to 50° on either side of the vertical. By loosening the clapper box lock screw, the clapper head can be swiveled on the tool slide in either direction to provide proper tool clearance on the return stroke when making angular or vertical cuts.

(2) In taking such cuts, always swivel the top of the clapper head in a direction away from the surface being machined, or the cutting tool will drag on the return stroke. In taking horizontal cuts, set the tool slide and clapper head vertically and feed the tool down into the work by means of the tool slide handle. In taking angular cuts, as when shaping a dovetail or a V-slot, set the tool slide at the same angle as the desired finished surface; the tool slide handle will then feed the cutting tool down at the correct angle. Between the tool slide and the ram itself, there is a scale graduated in degrees which permits angular settings to be measured accurately.

(3) The clapper box is a device which causes the cutting tool to lift from cutting position during the return stroke of the ram.

(4) The tool post is similar to those used on lathes; it will hold solid forged tools or a tool holder which in turn holds a suitable cutter bit.

40. Cutting tools.—*a. General characteristics.*—Shaper tools are, generally, the same as those used in lathes; however, solid forged tools are best for heavy cuts in large shapers.

b. Types of shaper tools.—The types of shaper tools in common use are roughing, finishing, side roughing, side finishing, parting, corner, radius, and special tools for specific operations such as T-slot cutting. All are available for right- or left-hand cuts and all can be used in either straight or offset tool holders.

c. Grinding.—Beware of dull tools. Only a carefully and properly ground tool will give satisfactory results in a shaper. The included angle of the cutting edge of a shaper tool should be the same for various metals as that of a similar lathe tool and the side clearance angle can be similarly ground. The heel clearance, however, which corresponds to the front clearance of a lathe tool should never be more than 3° for shaper work.

41. Operating the shaper.—*a. Holding the work.*—In general, shaper jobs are clamped or held in the same way as milling machine work (par. 33). Comparatively small pieces of regular shape to be machined in the shaper are usually held in a swivel vise or clamped directly to a rotary table. Larger work can be clamped directly to the top or side of the machine table. The type of swivel vise ordinarily used for holding shaper work has flat, parallel jaws as in figure 129. It is secured to either the top or side of the machine table by a clamp bolt and a removable tapered key. The jaws can be set at any angle in a plane horizontal to the cutting tool's direction of travel. Work larger than the vise will hold can be clamped directly to the top or side of the machine table as in milling machine work (par. 33*b*). The V-slot on the side of the table is convenient for holding round stock to be machined on the end. If the work must be rotated, but is too large or too awkward for a swivel vise, clamp it to a rotary table similar to the one shown in figure 132. Remember that in shaper set-ups, all vise jaws, clamps, clamp bolts, or other holding devices must be very tight. If the work becomes unseated during shaping, the cutting tool probably will be broken.

b. Speeds and feeds.—(1) Most shaper rams have three or four cutting stroke speeds regulated either by cone pulleys or by back gears. Since cutting speed is usually given in feet per minute, it must be converted into the corresponding speed in strokes of the ram per minute. To make this conversion, multiply feet per minute by 12 and divide by the length of the stroke in inches. Example: Find the number of strokes per minute to give a cutting speed of 35 feet per minute with a stroke of 6 inches.

$$35 \times 12 \div 6 = 70 \text{ strokes per minute.}$$

(2) Table XIX gives recommended speeds for roughing and finishing shaper cuts on various materials. It is important to use a finishing tool for finishing cuts in a shaper. Using the recommended speed and a light feed allows the tool to take a shearing cut which gives the work a good finish. As a rule, do not leave more than 0.010 to 0.012 inch of metal to be removed by finishing cuts.

(3) The feeds usually available on a shaper are the table cross feed operated either by hand or by power; and a limited down feed of the cutting tool operated by hand with the tool slide handle. The down feed of the cutting tool determines the depth of cut, which is

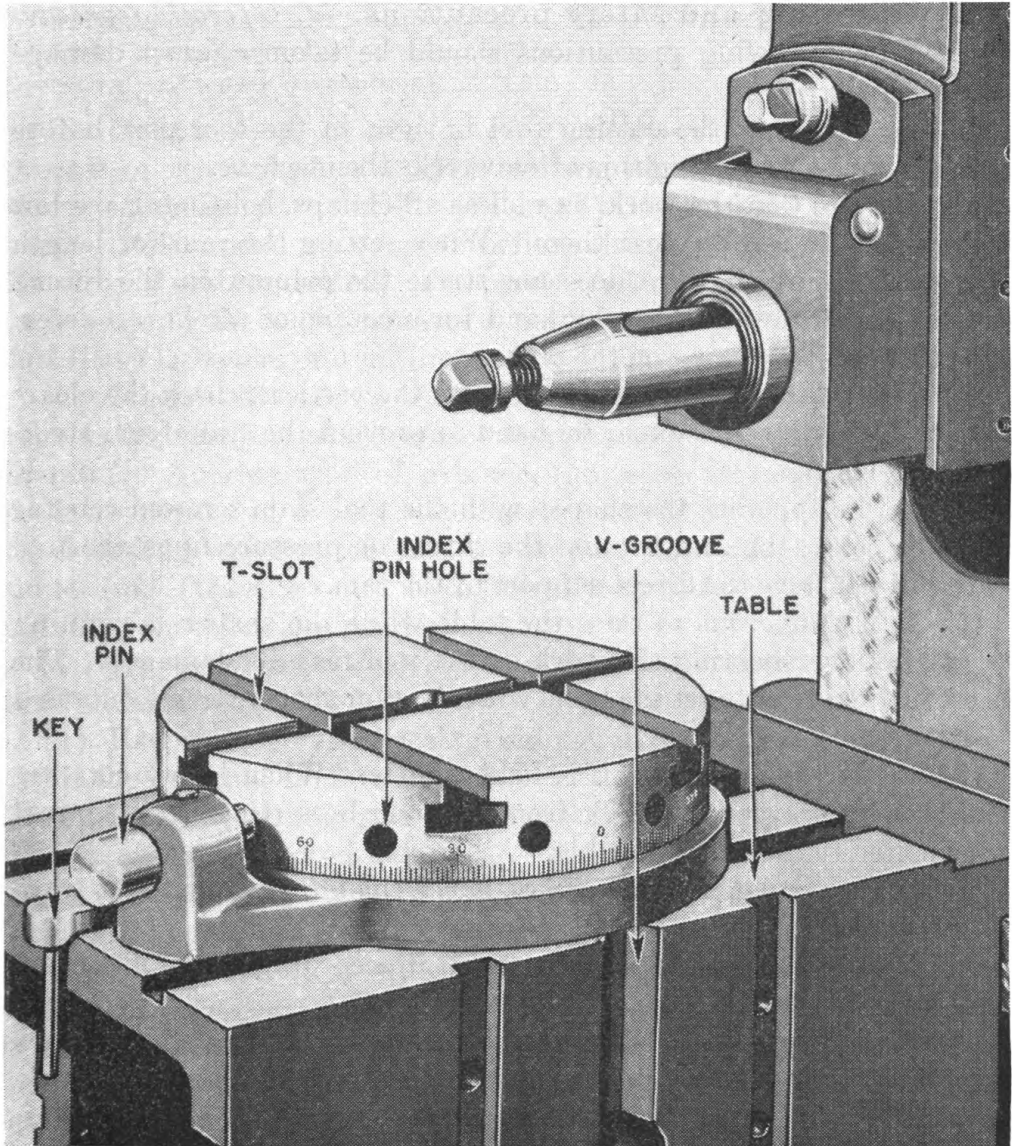


FIGURE 132.—Rotary shaper table.

generally $\frac{1}{32}$ to $\frac{1}{16}$ inch on small and medium size machines. (Cuts as deep as $\frac{1}{8}$ inch can be taken with heavy duty shapers.) The table cross feed occurs during the return stroke of the cutting tool so that the work is stationary while being cut. The cross feed should be as great as the type of cut being taken and the cutting tool being used

will stand. It should not, however, be so great that the tool will leave uncut ridges between strokes.

c. Coolants.—Table XIX specifies the proper coolants for shaper operations. Apply coolant liberally and keep the work, and whatever device is used to hold the work, free from chips.

42. Operating and safety precautions.—*a. Operating precautions.*—The following precautions should be taken against damage to the shaper:

(1) Be sure that the cutting tool is tight in the tool post before taking a cut. Neglecting this often wrecks the machine.

(2) Be sure that the work, as well as all clamps, bolts, etc., are low enough for the ram to clear them. When setting the ram for length of stroke be sure the ram does not strike the column on the return stroke. Operate the shaper by hand for a couple of strokes to determine this before turning on the power.

(3) When the tool slide is set off from the vertical, check the clearance by revolving the shaper by hand to prevent the slide from striking the column.

(4) Do not operate the shaper with the tool slide screwed too far above or below the ram because the thrust or pressure upon the tool slide will not have the direct support of the ram.

(5) It is dangerous to raise the table while the shaper is running because by overshooting the work or the tool may be damaged. The safest procedure is to set the table with the ram at its extreme outward position.

(6) Be sure that the work is held tightly. When heavy cuts are being taken, prevent the work from slipping by setting it up against stops bolted to the machine table.

(7) If the shaper has back gears, *never* change gears without stopping the machine.

(8) Before using a shaper, always oil it according to the manufacturer's directions.

b. Safety precautions.—The most common accident in shaper work is the fingers or hands being caught between the tool and the work. Keep the hands away from the machine table while the shaper is cutting, and never try to reach across the table between strokes of the tool. Because chips often fly from work being shaped, it is good practice always to wear goggles when operating a shaper.

SECTION VI

GRINDERS AND GRINDING

	Paragraph
General.....	43
Grinding wheels.....	44
Wheel selection.....	45
Grinding operations.....	46
Operating and safety precautions.....	47

43. General.—*a. Principles of grinding.*—(1) Grinding is the process of removing metal by a rotating abrasive wheel containing sharp, irregular particles of very hard material held together by a bond. It is a common error to believe that grinding wheels remove metal by a rubbing action. The process is that of *cutting*. Although each grain of the wheel removes a very small portion of the metal, it cuts like a lathe tool or a milling cutter.

(2) Figure 133 shows an 18 by 66-inch hydraulic crankshaft regrinding machine which can be used for any other type of external grinding. In this type of grinder, the work rotates on centers or eccentrically as in a lathe, and moves slowly from side to side as the grinding wheel is fed into it (see par. 16). The wheel feed and table feed (traverse) are hydraulically operated. Other types of grinding machines are similar.

b. Types of grinders.—Grinders and grinding operations are generally classified as—

(1) External or cylindrical grinding, in which metal is removed from the external surface of cylindrical or tapered work.

(2) Internal grinding, in which metal is removed from the walls of a straight or tapered hole.

(3) Surface grinding, in which metal is removed from a flat horizontal surface.

(4) Face grinding, in which metal is removed from a flat vertical surface.

(5) Centerless grinding, a comparatively new type of operation, in which the work is supported at two points on its surface and ground at another point opposite the points of support. It is a form of external grinding.

(6) Special grinding, which includes any grinding operation not previously classified. It includes honing, lapping, etc.

c. Range of operations.—Better finished work within closer tolerances can be obtained by grinding than by any other method. Bear in mind that grinding is primarily a finishing operation; general shop practice is to bring the work close to finished size by turning,

milling, shaping, or some other method, and to remove the small remaining portion by grinding. Hardened steel can be ground but cannot be worked with cutting tools. Many steel parts are turned

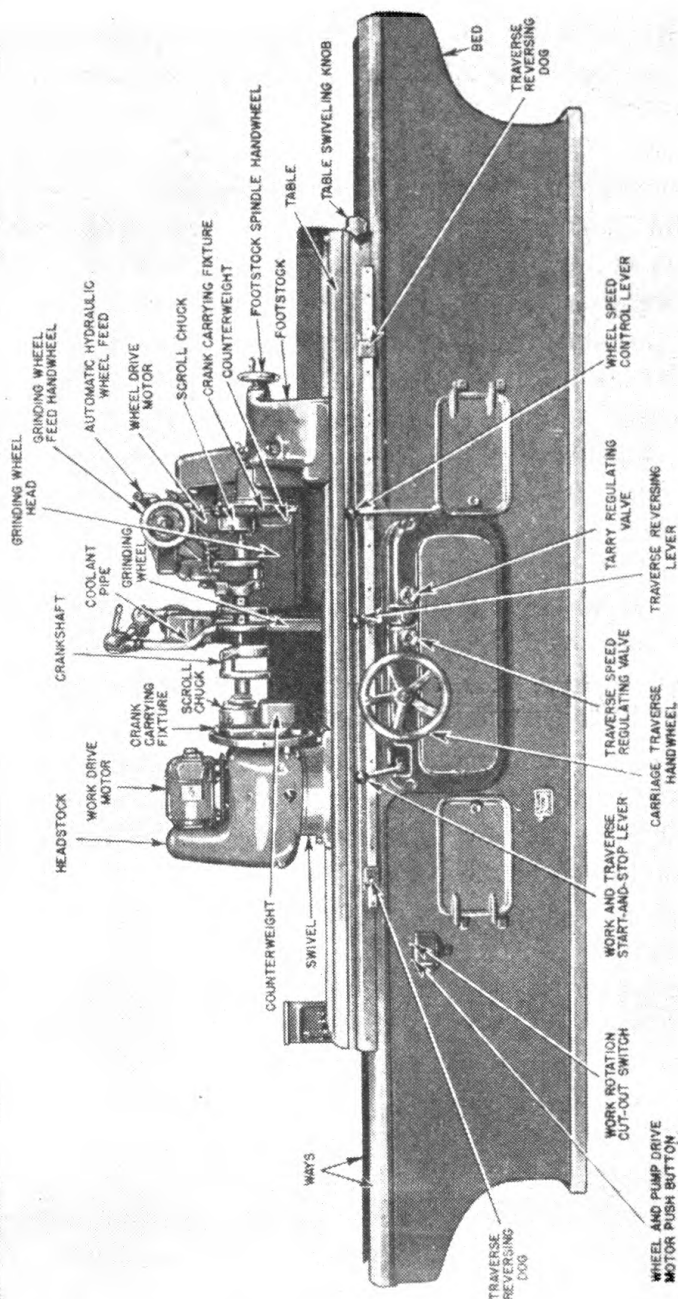


FIGURE 133.—Hydraulic crankshaft regrounding machine.

or milled in the annealed condition to approximate size, then hardened and finished to exact size by grinding. In the automotive repair shop, crankshafts, cylinder blocks, pistons and wrist pins,

valve faces and stems, rear axles, drive shafts, steering knuckle pivots, etc., are commonly finished on grinding machines.

44. Grinding wheels.—*a. Materials.*—Common grinding wheel abrasives are natural or manufactured. The minerals, emery and corundum, are the natural abrasives; manufactured abrasives are either aluminum oxide or silicon carbide sold under various trade names. Manufactured abrasives are the more widely used because the electric furnace process by which they are produced makes them uniform. Abrasives are graded according to the mesh of a sieve through which they are sorted. For example, grain No. 40 indicates that the abrasive passed through a sieve having 40 meshes to the linear inch.

b. Bonds.—Abrasive wheel bonds are classified as vitrified, silicate, or elastic.

(1) Vitrified wheels, most generally used, are bonded by earth or clay which hardens when heated to about 2,500° to 2,800° F. for a considerable length of time.

(2) Silicate wheels, bonded by silicate of sodium, are efficient for grinding cutting edges.

(3) Elastic wheels are bonded by a material made principally of shellac. Generally very thin, they are used for slitting and cutting-off operations.

c. Grades.—Abrasive wheels are generally graded as hard or soft according to the hardness of the bond or the strength with which it holds the abrasive. Manufacturers specify the grades of their wheels by numbers or letters, usually with the lowest numbers or letters identifying soft wheels (1, 2, etc., or A, B, etc.) and higher numbers or letters for harder wheels. Elastic wheels, for example, are graded as follows: 1, 1½, 2, 2½, 3, 4, 5, and 6. Grade 1 is the softest, grade 6 the hardest.

d. Shapes.—Figure 134 shows the nine standard shapes of grinding wheels. Other shapes, known as nonstandard, can be supplied on order. Figure 135 shows the standard shapes of faces that can be supplied on the above wheels; each is identified by the letter with which it is marked.

e. Mounting abrasive wheels.—(1) A wheel should get easily upon the machine spindle, but not loosely, or it cannot be centered accurately and is consequently out of balance. If the hole is too large, wrap paper around the spindle to make the wheel fit; if it is only slightly undersized, enlarge it with a file or, if it is leadbushed, with a jackknife. A wheel may easily be cracked by forcing it on a spindle, so care must be exercised.

(2) The flanges between which the wheel is mounted should be at least one-third, and preferably one-half, of the wheel diameter. They should be relieved on their inner sides so that only their outer edges bear against the wheel. This holds the wheel more securely with less pressure and with less danger of breaking it. The inner

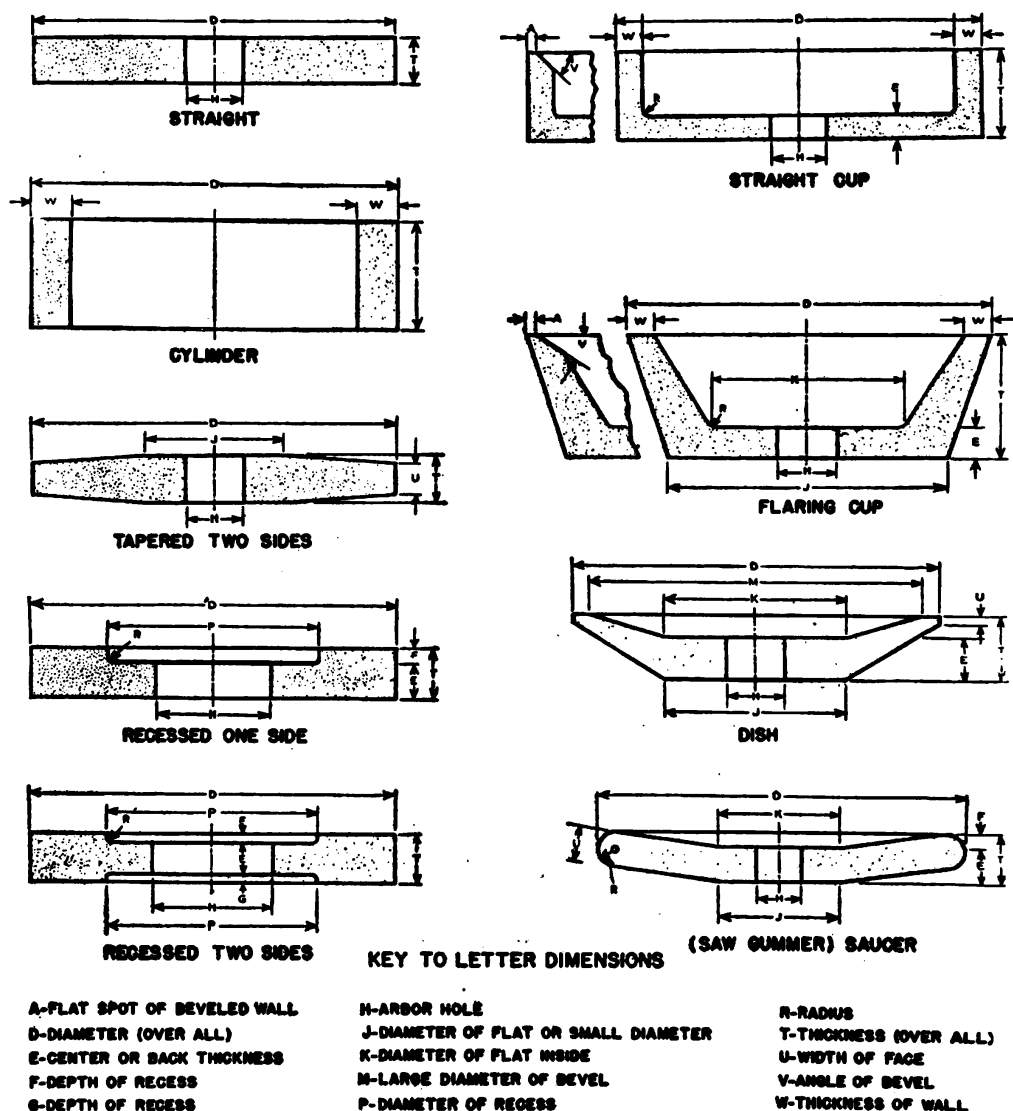


FIGURE 134.—Standard shapes of grinding wheels.

flange in a properly designed machine is keyed or otherwise fastened to the spindle to prevent the flange from turning and loosening the clamping nut.

(3) Leather, blotting paper, or rubber washers should be placed between the bearing surfaces of the flanges and the wheel.

(4) Tighten the clamping nut only enough to hold the wheel firmly or the clamping strain is likely to damage the wheel or parts of the machine.

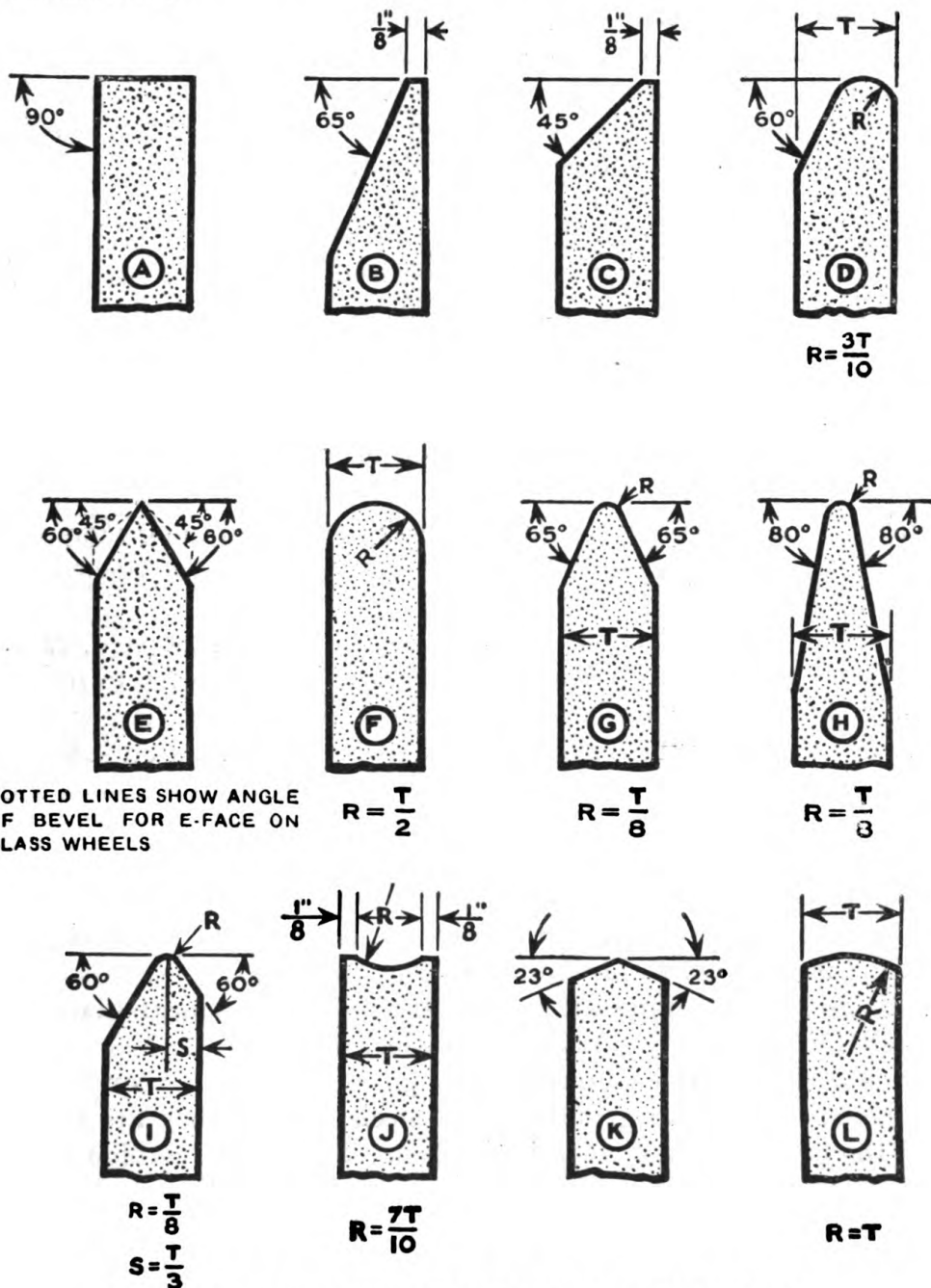


FIGURE 135.—Standard face shapes for standard grinding wheels.

f. Dressing the wheels.—Abrasive wheels for precision grinding machines are trued with the hardest known substance, a diamond, set in the end of a holder. The diamonds used for this purpose,

which are small, black, and rough, and therefore cheaper than gems, are commonly known as "industrial diamonds." They wear very slowly. The diamond tool is held on the machine so it travels across the face of the wheel. Always be sure that the diamond is firm in the holder. The diamond tool should be tilted about 5° up from the horizontal diameter of the wheel to wipe across its face rather than dig into it. Moving the diamond too rapidly across the face cuts a spiral or thread in the wheel which will produce a mottled or frosted appearance on the work and spoil the finish. Take light cuts with the diamond, cutting away no more of the wheel than is necessary to true its surface. Do not feed the diamond more than 0.001 inch per traverse. Allow plenty of coolant to flow on the diamond while truing.

45. Wheel selection.—*a.* Ordinarily, softer wheels are used for harder material and harder wheels for softer material. Hard materials require the sharpest cutting edges. Therefore a softer wheel, in which the particles will break away more easily under the cutting pressure as they become slightly dulled, exposing new sharp particles, is used in grinding hard materials, such as hardened steel and cast iron.

(1) Unhardened carbon steel and untreated machine steel can be ground to an excellent finish with cutting points too worn to produce an acceptable finish on harder materials; therefore a harder wheel that will retain its cutting particles longer may be used on soft steel.

(2) Still softer materials, such as brass, bronze, copper, and hard rubber, exert less pressure against the wheel than soft steel; therefore a soft wheel must be used or the cutting particles will be retained too long.

(3) The narrower the line of contact between the work and the wheel, the harder the wheel must be.

(4) Soft or medium-soft wheels are more economical than hard wheels for almost any grinding operation.

b. Aluminum-oxide abrasives, though softer than silicon carbide, are better adapted to grinding most steels because they are tougher and the grains are not so easily broken. The silicon carbide grains, which break easily when used on steel, cause rapid wheel wear. A general rule is to use silicon carbide abrasives only for materials of low tensile strength regardless of their hardness, and to use aluminum oxide for materials of high tensile strength, such as steel. Below is given a list of materials suggested for grinding with the two types of wheels:

<i>Silicon carbide</i>	<i>Aluminum oxide</i>
Gray and chilled iron	Carbon steels
Brass and soft bronze	Alloy steels
Aluminum and copper	High-speed steels
Rubber and leather	Annealed malleable iron
Very hard alloys	Wrought iron
Cemented carbides	Rough and hard bronzes

c. Although these general rules should prove helpful, the simplest way to select a wheel is to consult the tables furnished by the various manufacturers. Since the designation of grit and hardness varies among the different manufacturers, it is impossible to include such a table in appendix I.

46. Grinding operations.—*a. Grinding allowance.*—If the grinding machine is modern in design, leave as much as $\frac{1}{32}$ inch, or even more, on large machine-steel parts to be removed by the grinder but generally not more than $\frac{1}{64}$ inch on small machine parts.

b. Speeds and feeds.—(1) *Wheel speeds.*—Surface speeds of 5,000 to 6,000 feet per minute are most commonly used. Vitrified and silicate wheels should not be operated faster, except on cylindrical machines, where a speed of 6,500 fpm may be employed. Harder bakelite and rubber wheels may be operated up to 9,000 fpm. Do not confuse revolutions per minute with feet per minute. For a given rpm, a larger wheel will have a greater surface speed (fpm).

(a) *Rule I.*—To obtain surface speed when wheel diameter and revolutions per minute are given, multiply revolutions per minute by one-fourth the diameter of the wheel in inches, as

$$\text{Surface speed (fpm)} = \text{rpm} \times \frac{1}{4} \text{ diameter}$$

(b) *Rule II.*—When the diameter of the wheel in inches is given and the number of revolutions per minute necessary to give the desired cutting speed is needed, divide the required cutting speed in feet per minute by one-fourth the diameter of the wheel as

$$\text{rpm} = \frac{\text{cutting speed (fpm)}}{\frac{1}{4} \text{ diameter}}$$

Adjusting the speed may compensate for excessive hardness or softness in a wheel. If a hard wheel glazes because the dulled particles have not broken away, reduce the wheel speed or increase the work speed. If a soft wheel breaks down too rapidly, reverse the procedure.

(2) *Work speeds.*—These depend on the size and the nature of the material and on whether it is rigid enough to hold its shape. The larger the diameter of the work, the greater its arc of contact with the

wheel. The surface speed suitable for one diameter of work might be unsuitable for another. The fastest work speed that the machine and wheel will stand should be used for roughing.

(a) The following cylindrical work speeds are only typical: Steel shafts, 50-55 fpm; hard steel rolls, 80-85 fpm; chilled iron rolls, 80-200 fpm; cast iron auto pistons, 150-400 fpm; auto crankshaft bearings, 45-50 fpm; auto crankshaft pins, 35-40 fpm.

(b) Surface grinding machines are sometimes built with a constant table speed of 80 feet per minute.

(c) Traverse feeds are set according to the width of the wheel. In roughing, the work should cross $\frac{3}{4}$ to $\frac{7}{8}$ of the wheel's width each time the work revolves. In finishing, use a finer feed, generally $\frac{1}{8}$ to $\frac{1}{4}$ of the width of the wheel for each revolution of the work.

(3) *Depth of cut.*—In the roughing operation the cut should be as deep as the wheel will stand without crowding, depending upon the hardness of the material and the diameter of the work. Experience is the only guide. In the finishing operation the depth of cut is always slight, from 0.0025 to 0.001 inch. An excellent finish is obtained by letting the wheel run over the work several times without cross feeding. This practice of letting the wheel "grind out" yields results satisfactory to most expert operators even with a comparatively coarse wheel.

c. Coolant.—In most grinding machines a steady flow of cutting lubricant or coolant must be directed on the part of the work where the wheel touches. Uneven temperature distorts the work. The coolant prevents this, keeps the wheel clean and free cutting, and puts a better finish on the work. Some commercial brand of soluble oil, mixed with water, is generally used.

d. Testing work for accuracy.—Accurate grinding requires fine measuring tools. A micrometer caliper, especially with vernier graduation, is well suited for this work. The mechanical principles, construction, use, and care of micrometers are described in detail in paragraphs 28 to 32, inclusive, TM 10-590. Become proficient in measuring with micrometer calipers before undertaking grinding operations.

e. Grinding in Army motor maintenance.—The commonest grinding operations in Army motor maintenance are: piston grinding, crankshaft grinding, cylinder grinding, and clutch plate grinding.

(1) Piston grinding is an adaptation of ordinary external cylindrical grinding. Pistons cannot be ground between centers since the skirt end of the piston is hollow; therefore a conical adapter is used on the grinding machine for centering this end, while an ordinary dead center is used for the head end. The ring lands are undercut

to some specified diameters less than the skirt diameter. Pistons are usually of aluminum alloy but sometimes cast iron. Different grinding wheels should be used on each of these materials for best results. Pistons must be ground to fit the cylinders in which they are to work; that is, the size of the cylinder bore, less the specified clearance. If the best practice has been followed in the cylinder reconditioning, however, the cylinders will all be the same standard oversize, such as 0.010, 0.020, 0.030, 0.040, or 0.050 inch.

(2) Main journals of crankshafts are ground between centers like any other plain cylindrical surfaces, with center rests to prevent them from being sprung by the pressure of the grinding wheel. They are preferably ground to standard undersizes in multiples of 0.010 inch to simplify fitting the bearings. Crankpins are ground with special adjustable fixtures (fig. 133) which center them by holding each end of the crankshaft in an eccentric position. A dial indicator is generally used to determine when the pin is centered. The rest is placed against the pin being ground. A portable crankshaft grinder (fig. 136) is designed primarily to be mounted on the carriage of an ordinary lathe, thereby converting it into a crankshaft grinder for main bearings and crankpins. It can be attached to the bottom of an engine block to grind the crankpins while the crankshaft is in the engine. In this case an electric motor rotates the crankshaft through one of the vehicle rear wheels. A hook-shaped work follower rest causes the wheel to follow the crankpin around in its orbit, the pull of the hook working against spring tension on the rocker yoke. This arrangement makes it unnecessary for the crankshaft to be mounted eccentrically while the crankpins are being ground. The follower rest meets the crankpin at two points opposite the grinding wheel, making this method an adaptation of the principle of centerless grinding. The machine feeds the cup wheel horizontally across the journal and vertically into it and has an eccentric dial for correcting taper in the work.

(3) Cylinder grinding is done on a special internal grinding machine. A motor block is too large and heavy to be rotated; therefore it is set up in a stationary position on the machine. A small grinding wheel revolves on a spindle which in turn rotates in an orbit. Thus the grinding wheel follows the cylinder wall around with a planetary motion. The cylinder grinder is now used very little in Army motor maintenance. The usual practice is to rebore the cylinders to standard oversizes with a portable cylinder boring machine and finish with a hone, described in paragraph 38, TM 10-590.

(4) Clutch-plate grinding is an application of surface grinding done on an ordinary surface grinding machine or a special machine designed especially for the purpose.

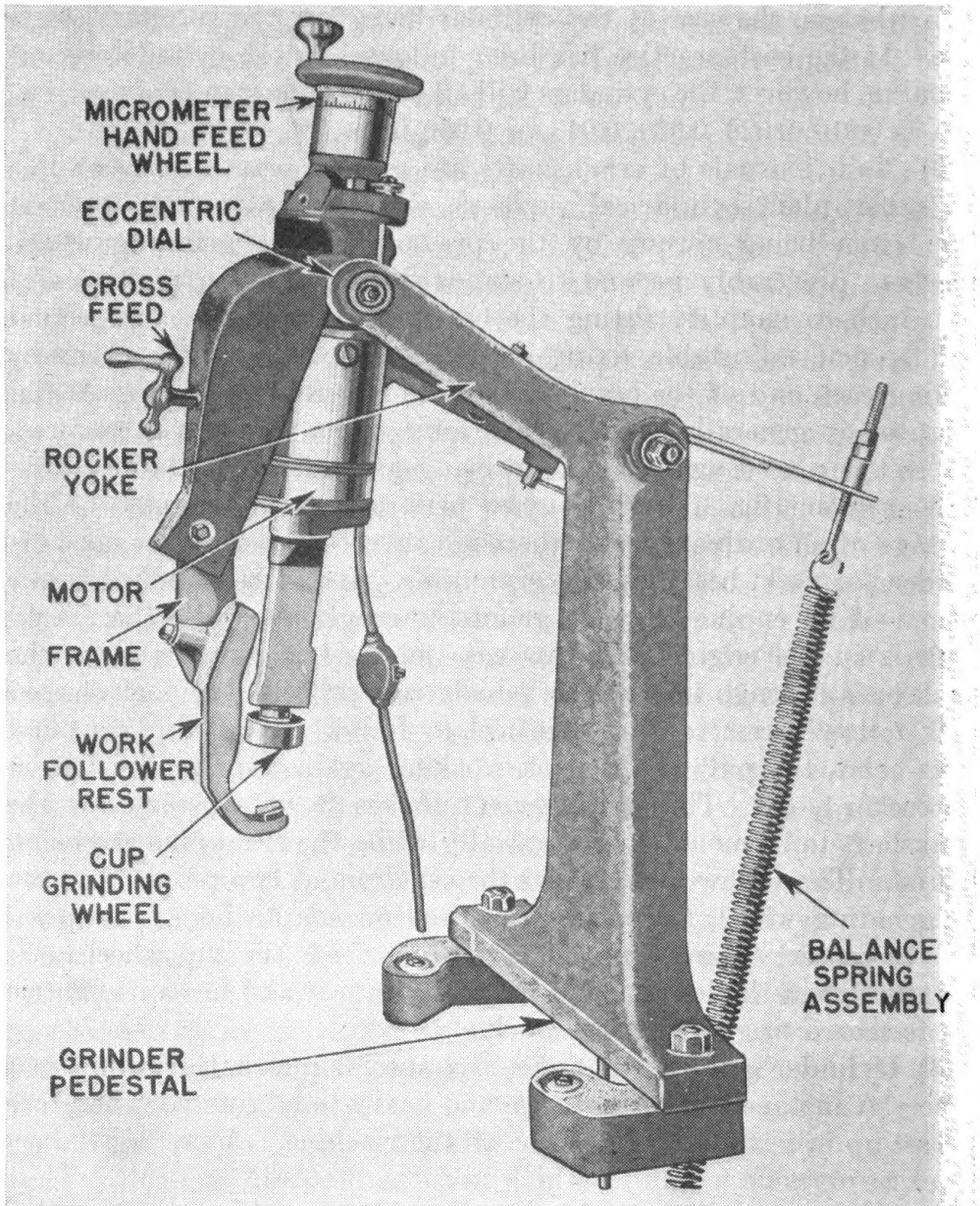


FIGURE 136.—Portable crankshaft grinder.

f. Lapping.—(1) In certain lines of work the final grinding process is often accomplished not with abrasive wheels but with metal disks, rings, or cylinders, the surfaces of which have been charged with a

fine flour abrasive. Such a tool, called a lap, finishes work to the most accurate fits and dimensions possible in a process called lapping.

(2) A lap is generally made of some material soft enough that the abrasive can be readily pressed into its surface. This process is known as charging the lap. Soft, close-grained cast iron, copper, brass, or lead may be charged with any of the flour abrasives by using a hardened roller or rolling them in the flour on a hardened surface. Since lapping is a somewhat slow and tedious process it should be used for the removal of only very small amounts of stock.

(3) Lapping is commonly used to finish micrometer ends, plug and ring gages, holes in jig bushings, and the finest die and punch work. The only common application today in Army automotive maintenance is to seat valves, which serve as the laps. Although piston rings are usually purchased in widths to fit the piston grooves exactly, they sometimes must be dressed down in width by lapping. To do this, rub the ring by hand on a piece of abrasive cloth laid flat on a surface plate. Formerly cylinder and piston walls were often finished by hand lapping, but today sufficiently fine finishes are obtained by mechanical grinders and hones.

47. Operating and safety precautions.—*a.* Chatter in grinding is any deviation from a smooth surface. Any number of things may cause chatter. The most frequent causes are an improperly balanced or insecurely held grinding wheel, improper grade of wheel, or loose bushing fit on the spindle. Additional causes include improper angle of the work center point, misfit of work centers in spindles, loose headstock or footstock spindles, poor bearing surfaces on work carriage or wheel slide, no backlash tension on wheel head, and insufficient number or improper use of work rests. To overcome chatter, check each possible cause patiently and persistently.

b. Frequently the work ground is out of round. This may result when the work is out of balance, center holes in the work are out of round or have an improper angle, the work centers are badly worn, the wheel is too hard and fine, there is insufficient coolant, or the release of internal stresses strains the work. If plenty of water is kept flowing on the work and a good free cutting wheel is used, this trouble is usually avoided.

c. Occasionally a grinding machine will size inaccurately for no apparent reason. Even when the wheel is fed in repeatedly with the stop on the hand feed wheel set in the same position, it will grind one piece oversize, another piece the correct size, and still another piece undersize. Increasing the backlash tension on the wheel base usually corrects this condition.

- d.* Clean and oil all finished or polished surfaces of the machine before leaving it idle for any length of time.
- e.* Keep dirt and grit out of oil holes or bearings.
- f.* Clean the top of the swivel table before changing the position of the headstock or footstock.
- g.* Clean and oil the work centers before putting work in the machine.
- h.* Change wheels when necessary. One wheel for all classes of work is false economy.
- i.* If a wheel is crowded beyond a certain point it will not cut more rapidly but will simply heat the work and wear out the wheel.
- j.* Long work cannot be ground accurately without the proper number of work rests.
- k.* A little kerosene will reduce any undue foaming of the grinding compound.
- l.* When truing the wheel use plenty of water on the diamond.
- m.* It is not wise to tamper with the grinding wheel spindle bearings unless absolutely necessary.
- n.* Be sure that all grinding wheels are held firmly on their arbors. The inner flange should be keyed or otherwise fixed to the spindle; the outer flange should bear on the wheel at its outer edge through thick, soft paper or rubber, and the wheel should run true.
- o.* In every case, an abrasive wheel should be well guarded to prevent accidents from flying particles from a broken wheel.
- p.* It is not wise to force the grinding wheel on the spindle.
- q.* A grinding wheel that is not sound is dangerous.

SECTION VII

POWER HACKSAWS

Description.....	Paragraph 48
------------------	-----------------

48. Description.—*a.* The power hacksaw, a piece of equipment found in any but the smallest machine shops, is used for cutting off bar stock, pipe, tubing, or any metal stock within its capacity. It is much faster and easier than hand sawing and makes an accurate, square cut.

b. Ordinarily, power hacksaws range in capacity from stock 4 inches square up to stock 13 by 16 inches. The stroke of the saw varies from 4 inches in the smaller machines to 6 inches in the larger ones.

(1) Some machines feed by gravity, the saw frame being provided with weights that can be shifted to give greater or less pressure on

the saw. Other machines have a mechanical feed which ranges from 0.001 to 0.025 inch per stroke, depending on the class of material cut. The machine illustrated in figure 137 has a hydraulic feed. When hard spots are encountered, the feed is automatically stopped or shortened to decrease the pressure on the saw until the hard spot has been cut through. All machines lift the saw clear of the work on the return stroke.

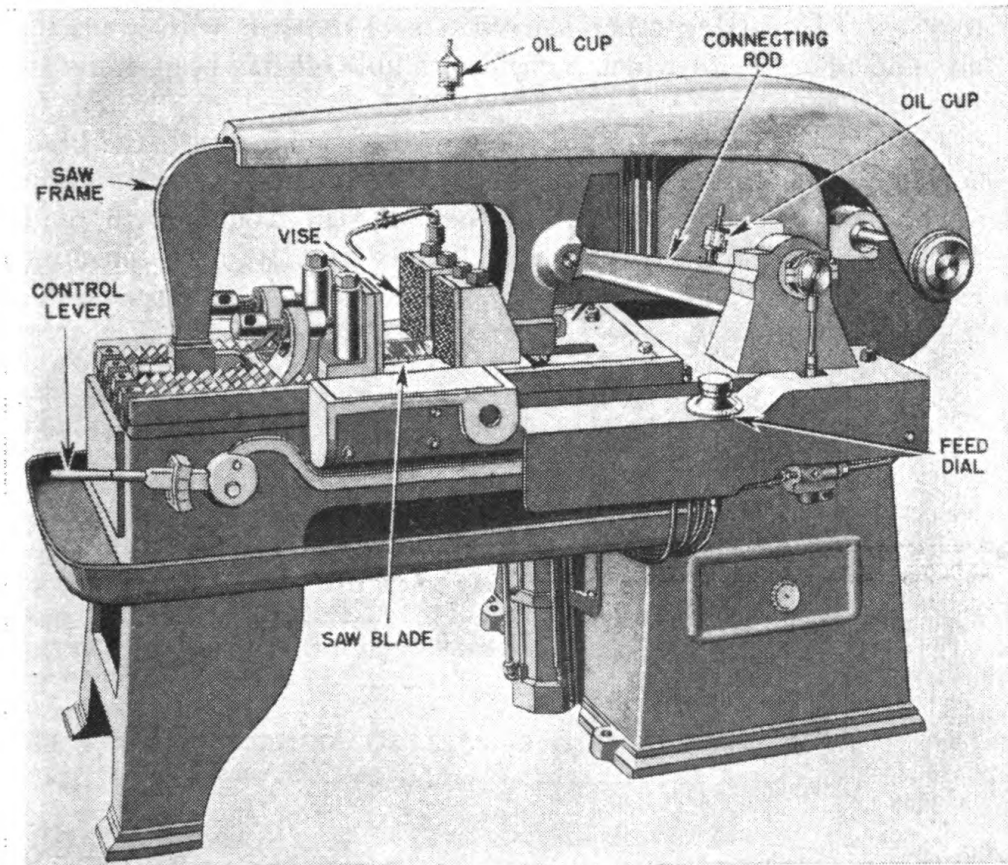


FIGURE 137.—Power hacksaw.

(2) Hacksaw machines are both belt and motor driven. Motors range from $\frac{1}{4}$ hp for a 4 by 4 inch machine to 2 hp in the largest size. Heavy duty machines have three speeds.

c. Blades for power hacksaws are usually manufactured in 12-, 14-, 17-, 18-, 21-, and 24-inch lengths. There are four general classes of teeth which are used principally as follows:

Regular (4 per inch) for soft steel, cast iron, bronze; *medium* (6 per inch) for annealed high-carbon steel, high-speed steel; *fine* (10 per inch) for solid brass, iron pipe, heavy tubing; *tubing* (14 per inch) for thin tubing, sheet metals. Choose the pitch of teeth ac-

ording to the size of the section being cut. For wide, heavy sections, use coarse-pitch teeth to give ample chip clearance. For thin sections, choose a pitch that keeps two or more teeth in contact with the work; otherwise the teeth will straddle the work and strip the saw.

d. One prominent manufacturer recommends the following speeds in strokes per minute: For cold-rolled or machine steel, brass, soft metals, 136; alloy steel, annealed tool steel, cast iron, 90; high-speed steel, unannealed tool steel, stainless steel, 60. Cast iron should be sawed dry. In cutting other materials, cool the saw with a cutting compound of water to which sufficient soluble oil has been added to whiten it.

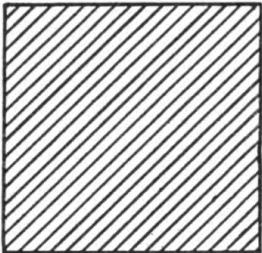
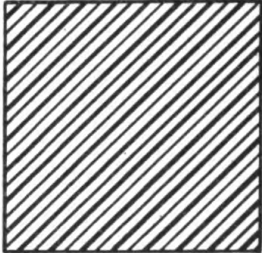
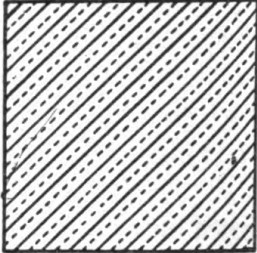
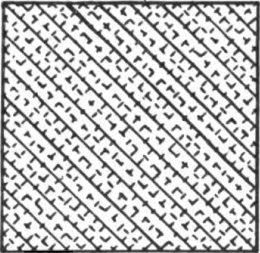
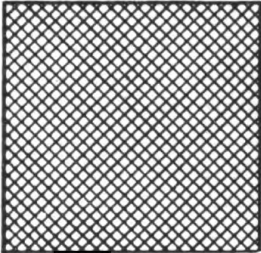
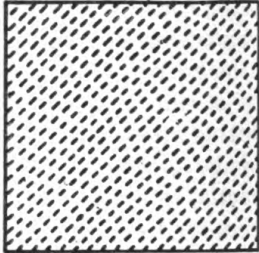
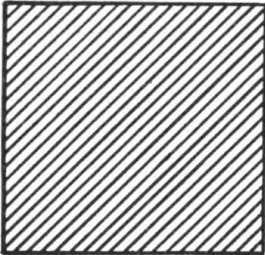
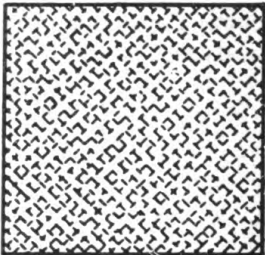
e. Often a blade is broken by starting the machine with the blade touching the work. Start the machine with the blade somewhat above the work and then feed it in. If there is still trouble with blade breakage, the work is probably not clamped tightly in the machine.

APPENDIX I

TABLES

	Table
Section lines or cross hatching.....	I
Conventional abbreviations.....	II
Decimal equivalents.....	III
Grinding twist drills.....	IV
Suggested speed and feed for high-speed steel drills.....	V
Drilling troubles, causes and remedies.....	VI
Speeds for dies and taps.....	VII
Reamer speeds.....	VIII
Lathe cutting speeds and lubricants.....	IX
Size of center hole for $\frac{3}{16}$ -inch to 4-inch diameter shafts.....	X
Cutting speeds for turning.....	XI
Spindle speeds in rpm for turning and boring.....	XII
Practical sizes of chucks for lathes.....	XIII
Rules for figuring tapers.....	XIV
American national standard screw thread pitches and recommended tap drill sizes.....	XV
American national special screw thread pitches (N. S.) and recommended tap drill sizes.....	XVI
Thread-cutting data.....	XVII
Milling machine cutting speeds and lubricants.....	XVIII
Shaper cutting speeds and feeds.....	XIX

TABLE I.—*Section lines or cross hatching*

		
CAST IRON	STEEL	COPPER
		
BRASS OR BRONZE	WHITE ALLOYS	ALUMINUM
		
LEAD	ZINC	

THE MACHINIST

TABLE II.—*Conventional abbreviations*

All information on a drawing is, when possible, abbreviated as follows:

Abbreviation	Meaning
Bore.....	Use of boring tools or bars.
C. I.....	Cast iron.
C. R. S.....	Cold rolled steel.
C. S.....	Carbon steel.
Dia.....	Diameter.
Drill.....	Hole is to be drilled.
Face.....	To square up.
Finish.....	Surface is to be finished.
Grind.....	Surface is to be ground.
H. S. S.....	High-speed steel.
L. H.....	Left hand.
M. S.....	Machine steel.
Rad.....	Radius.
Ream.....	Hole should be reamed.
R. H.....	Right hand.
Running fit, drive fit, force fit, shrink fit, taper fit.	Allowances to be made in size of shaft.
Scrape.....	Surface is to be handscraped.
Tap.....	Hole is to be tapped.
Thd.....	Thread.
T. S.....	Tool steel.
U. S. S.....	United States standard.
W. I.....	Wrought iron.
Symbols:	
'.....	Feet.
".....	Inches.

Screw threads, structural riveting, pipe fittings, lineshaft bearings, etc., are so standardized that the above abbreviations are always used by draftsmen.

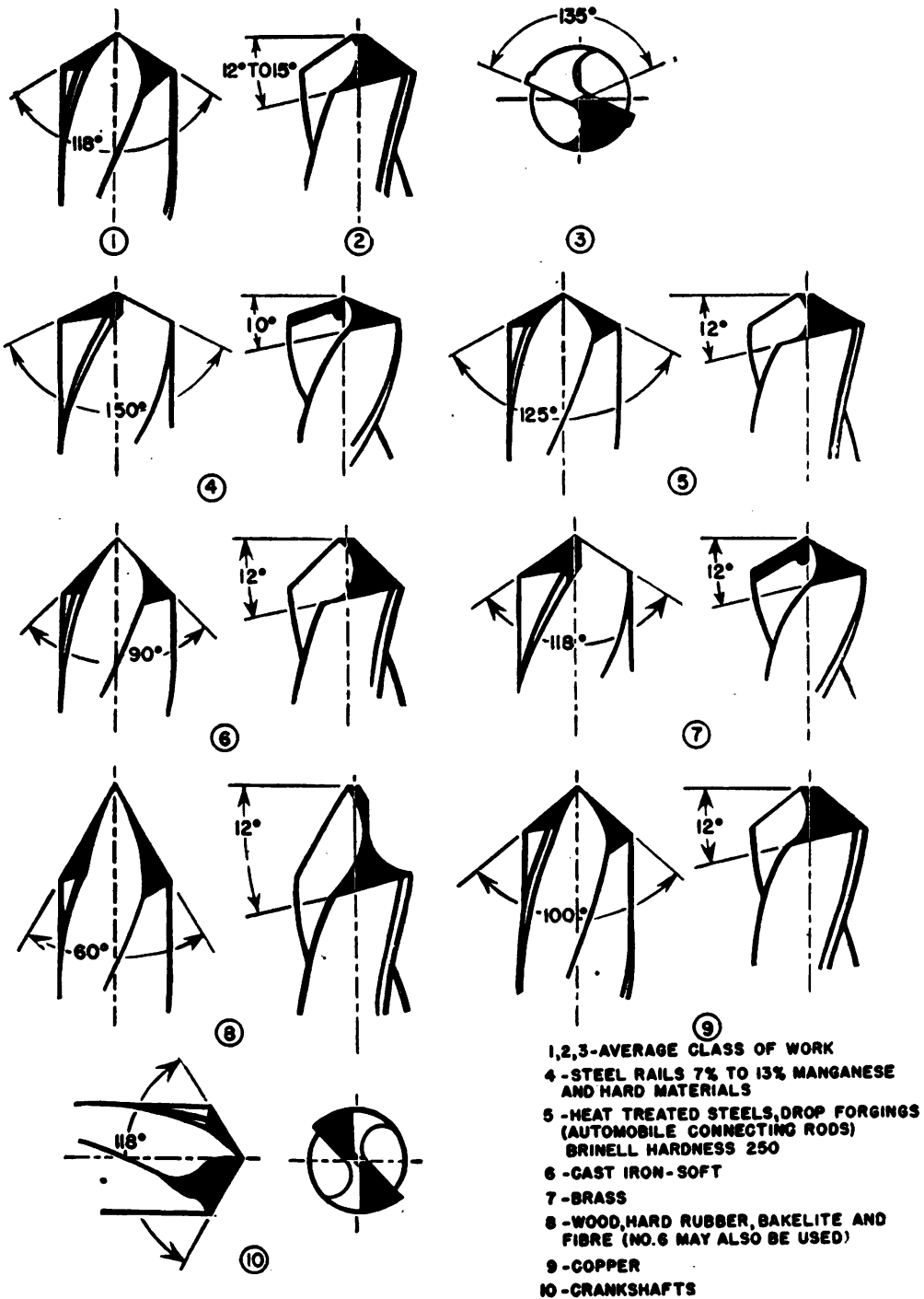
QUARTERMASTER CORPS

TABLE III.—*Decimal equivalents*

Fraction	Decimal equivalent	Fraction	Decimal equivalent	Fraction	Decimal equivalent	Fraction	Decimal equivalent
$\frac{1}{64}$ -----	0. 015625	$\frac{17}{64}$ -----	0. 265625	$\frac{33}{64}$ -----	0. 515625	$\frac{49}{64}$ -----	0. 765625
$\frac{1}{32}$ -----	. 03125	$\frac{9}{32}$ -----	. 28125	$\frac{17}{32}$ -----	. 53125	$\frac{25}{32}$ -----	. 78125
$\frac{3}{64}$ -----	. 046875	$\frac{19}{64}$ -----	. 296875	$\frac{35}{64}$ -----	. 546875	$\frac{51}{64}$ -----	. 796875
$\frac{1}{16}$ -----	. 0625	$\frac{5}{16}$ -----	. 3125	$\frac{9}{16}$ -----	. 5625	$\frac{13}{16}$ -----	. 8125
$\frac{5}{64}$ -----	. 078125	$\frac{21}{64}$ -----	. 328125	$\frac{37}{64}$ -----	. 578125	$\frac{53}{64}$ -----	. 828125
$\frac{3}{32}$ -----	. 09375	$\frac{11}{32}$ -----	. 34375	$\frac{19}{32}$ -----	. 59375	$\frac{27}{32}$ -----	. 84375
$\frac{7}{64}$ -----	. 109375	$\frac{23}{64}$ -----	. 359375	$\frac{39}{64}$ -----	. 609375	$\frac{55}{64}$ -----	. 859375
$\frac{1}{8}$ -----	. 125	$\frac{3}{8}$ -----	. 375	$\frac{5}{8}$ -----	. 625	$\frac{7}{8}$ -----	. 875
$\frac{9}{64}$ -----	. 140625	$\frac{25}{64}$ -----	. 390625	$\frac{41}{64}$ -----	. 640625	$\frac{57}{64}$ -----	. 890625
$\frac{5}{32}$ -----	. 15625	$\frac{13}{32}$ -----	. 40625	$\frac{21}{32}$ -----	. 65625	$\frac{29}{32}$ -----	. 90625
$\frac{11}{64}$ -----	. 171875	$\frac{27}{64}$ -----	. 421875	$\frac{43}{64}$ -----	. 671875	$\frac{59}{64}$ -----	. 921875
$\frac{3}{16}$ -----	. 1875	$\frac{7}{16}$ -----	. 4375	$\frac{11}{16}$ -----	. 6875	$\frac{15}{16}$ -----	. 9375
$\frac{13}{64}$ -----	. 203125	$\frac{29}{64}$ -----	. 453125	$\frac{45}{64}$ -----	. 703125	$\frac{61}{64}$ -----	. 953125
$\frac{7}{32}$ -----	. 21875	$\frac{15}{32}$ -----	. 46875	$\frac{23}{32}$ -----	. 71875	$\frac{31}{32}$ -----	. 96875
$\frac{15}{64}$ -----	. 234375	$\frac{31}{64}$ -----	. 484375	$\frac{47}{64}$ -----	. 734375	$\frac{63}{64}$ -----	. 984375
$\frac{1}{4}$ -----	. 25	$\frac{1}{2}$ -----	. 5	$\frac{3}{4}$ -----	. 75	1-----	1.

THE MACHINIST

TABLE IV.—Grinding twist drills



1. General.—*a.* Few operations on tools in the shop are more frequently disappointing than the grinding or sharpening of drills.

b. To get satisfactory performance from a drill, the cutting edges must have a proper and uniform angle with the longitudinal axis of the drill. This angle is sometimes expressed as the angle between the cutting edge or lip, and the axis of the drill and sometimes as the angle included between the two lips, which is twice as much. The angles shown in table IV are included angles. They must be exactly equal in length with the proper lip clearance or contour of the surface back of the cutting edges. This clearance must be the same on each lip and the chisel point must have the proper angle.

c. The commercial standard for angle of drill point (118° included angle or 59° between the cutting lip and the axis, table IV①; 12° to 15° lip clearance, table IV②; angle of chisel point 125° to 135° , table IV③) is best suited for drills used in all average classes of work.

d. The form of a drill point exerts a powerful influence upon the rate of production, accuracy of drilled holes, and the number of holes which can be drilled between successive grindings.

2. Effect of improper point grinding and common drill faults.—*a.* Cutting lips which are not of uniform length will drill the hole oversize and the margin of one cutting lip will wear. If the drill has no lip clearance or is inclined to a negative rake, it will not cut, causing considerable stress on the machine and burning the heel of the drill.

b. Too much lip clearance, leaving an insufficient amount of metal behind the cutting lips, will chip and crumble cutting edges.

c. Too fast a peripheral speed breaks down the outer corners of the cutting lips and increases wear on the margins of the drill.

d. Too heavy a feed crushes the drill point and often breaks the drill.

e. An imperfectly fitted taper shank in the socket (due to dirt, chips, or worn-out socket) breaks the tangs.

3. Modifying factors.—*a.* In up-to-date practice, drilling different grades of materials requires at times a modification of the commercial 118° drill point for maximum results.

b. Hard materials require a blunter point, and soft materials a more acute angle.

c. A list of speeds and feeds for drills used in various materials which were established on a production basis is given in table V. Use it as a standard reference from which to deviate according to

THE MACHINIST

conditions. Variable factors involved in drilling operations, such as different materials, degrees of hardness, depth of holes, lack of uniformity of materials, and lubrication, make it difficult to establish a list of speeds and feeds that will conform satisfactory to all conditions.

TABLE V.—*Suggested speed and feed for high-speed steel drills*

Diameter of drill (inches)	Revolutions per minute											Feed per revolution (inches)
	Aluminum	Brass	Cast iron ¹	Mild steel 20-30 carbon	Steel 40-50 carbon	Tool steel 1.20 carbon and drop forgings	Conn. rod molybdenum steel	3.50 nickel steel	Stainless steel and monel metal	Malleable iron	Slate	
	300 feet	200 feet	100 feet	110 feet	80 feet	60 feet	55 feet	65 feet	50 feet	85 feet	15 feet	
1/16	18,336	12,224	6,112	6,724	4,883	3,668	3,404	3,976	3,056	5,192	916	.0015
1/8	9,168	6,112	3,056	3,362	2,444	1,834	1,702	1,988	1,528	2,596	458	.002-.003
3/16	6,108	4,072	2,036	2,242	1,630	1,222	1,120	1,324	1,018	1,734	306	.004
1/4	4,584	3,056	1,528	1,681	1,222	917	851	994	764	1,298	229	.005
5/16	3,666	2,444	1,222	1,344	978	733	672	794	611	1,039	184	.005
3/8	3,054	2,036	1,018	1,121	815	611	560	662	509	867	153	.006
7/16	2,622	1,748	874	921	699	524	481	568	437	742	131	.007
1/2	2,292	1,528	764	840	611	459	420	497	382	649	115	.008
5/8	2,037	1,358	679	747	543	407	373	441	340	577	102	.008
3/4	1,836	1,224	612	673	489	367	337	398	306	520	92	.009
7/8	1,665	1,110	555	611	444	333	300	360	273	472	84	.009
1	1,524	1,016	508	559	408	306	279	330	254	433	77	.010
1 1/16	1,422	948	474	521	379	284	261	308	237	403	71	.010
1 1/8	1,314	876	438	482	349	262	241	285	219	371	66	.011
1 1/4	1,221	814	407	448	326	244	224	265	204	346	61	.012
1 1/2	1,146	764	382	420	306	229	210	258	191	325	58	.013
1 3/8	1,077	718	359	395	287	215	197	233	180	305	54	.013
1 1/2	1,020	680	340	374	272	204	187	221	170	288	51	.014
1 5/8	966	644	322	354	258	193	177	209	161	274	48	.014
1 3/4	918	612	306	337	245	183	168	199	153	260	46	.015
1 7/8	873	582	291	320	233	175	160	189	146	248	44	.015
1 3/4	834	556	278	306	222	167	153	180	139	236	42	.015
1 7/8	795	530	265	292	212	159	146	172	133	225	40	.015
1 1/2	762	508	254	279	204	153	140	165	127	216	38	.015
1 7/8	732	488	244	268	195	146	134	159	122	207	37	.016
1 3/4	702	468	234	257	188	141	129	152	117	201	35	.016
1 11/16	678	452	226	249	181	136	124	147	113	192	34	.016
1 3/4	654	436	218	240	175	131	120	142	109	186	33	.016
1 13/16	630	420	210	231	168	126	116	137	105	179	32	.016
1 3/4	612	408	204	224	163	122	112	133	102	173	31	.016
1 15/16	591	394	197	216	158	118	108	128	99	168	30	.016
2	573	382	191	210	153	115	105	124	96	162	29	.016

¹ Soft grade 50 percent faster.

Carbon-steel drills should be run at speeds from 40 to 50 percent of those given above with approximately the same feed.

QUARTERMASTER CORPS

TABLE VI.—*Drilling troubles, causes and remedies*

Symptoms	Probable cause	Remedy
Breaking of drill.	Spring or backlash in press or work. Too little lip clearance. Too low speed in proportion to the feed. Flutes full of chips. Dull drill.	Test press and work for rigidity and alignment. Regrind properly. Increase speed or decrease feed. Sharpen drill.
Breaking down of outer corners of cutting edges.	Material being drilled has hard spots, scale, or sand inclusions. Too much speed. Improper cutting compound. No lubricant at point of drill.	Reduce speed. Use proper cutting compound and correct application.
Breaking of drill when drilling brass or wood.	Chips clog up flutes.	Increase speed. Use drills designed for these materials.
Broken tang.	Imperfect fit of taper shank in the socket—due to nicks, dirt, burs, or worn-out socket.	Get a new socket or ream old one to prevent recurrence.
Chipping of margin.	Oversized jig bushing.	Use proper size bushing.
Chipping of lip or cutting edges.	Too much feed. Too much lip clearance.	Reduce feed—see table V. Regrind properly.
Chipping or checking of a high-speed drill.	Heated and cooled too quickly while grinding or while drilling. Too much feed.	Warm slowly before using. Do not throw cold water on hot drill while grinding or drilling. Reduce feed.
Change in character of chips while drilling.	Change in condition of the drill such as chipping of cutting edge, and dulling.	Regrind drill properly.

THE MACHINIST

TABLE VI.—*Drilling troubles, causes and remedies*—Continued

Symptoms	Probable cause	Remedy
Hole too large.	Unequal angle or length of the cutting edges— or both. Loose spindle.	Regrind properly. Test spindle for rigidity.
Only one lip cutting or one large, one small chip.	Unequal length or angle of cutting lips or both.	Regrind drill properly.
Splitting up center.	Too little lip clearance. Too much feed.	Regrind with proper lip clearance. Reduce feed.
Rough hole.	Dull or improperly ground drill. Lack of lubricant or wrong lubricant. Improper set-up. Too much feed.	Regrind properly. Lubricate or change lu- bricant. Reduce feed.

TABLE VII.—*Speeds for dies and taps*

(Run high-speed steel tools 25 percent faster)

Diameter of thread	Screw stock		Brass rod		Diameter of thread	Screw stock		Brass rod	
	Feet surface speed	Revolu- tions per minute	Feet surface speed	Revolu- tions per minute		Feet surface speed	Revolu- tions per minute	Feet surface speed	Revolu- tions per minute
1/8-----	25	764	100	3,056	3/4-----	20	102	80	408
1/4-----	25	382	90	1,376	1-----	20	76	80	306
3/8-----	20	204	80	815	1 1/4-----	20	67	80	245
1/2-----	20	153	80	611	1 1/2-----	15	38	70	178
5/8-----	20	123	80	489	2-----	15	29	70	134

TABLE VIII.—*Reamer speeds*

REAMING SPEEDS AND FEEDS IN SCREW MACHINES

Diameter of reamer, in inches	Feed per revolution, in inches	Amount to remove on diameter, in inches ¹	Revolutions per minute at various peripheral speeds			
			Screw stock at 40 feet	Brass rod at 130 feet	Cast iron at 45 feet	Tool steel at 25 feet
$\frac{1}{8}$ -----	0. 005	0. 0045	1, 222	3, 972	1, 375	764
$\frac{3}{16}$ -----	. 006	. 0045	815	2, 648	917	509
$\frac{1}{4}$ -----	. 007	. 006	611	1, 986	688	382
$\frac{3}{8}$ -----	. 0085	. 006	407	1, 324	458	254
$\frac{1}{2}$ -----	. 0105	. 008	306	993	344	191
$\frac{5}{8}$ -----	. 012	. 008	245	795	275	153
$\frac{3}{4}$ -----	. 014	. 008	204	662	229	127
1-----	. 016	. 010	153	497	172	95
$1\frac{1}{4}$ -----	. 018	. 010	122	397	138	76
$1\frac{1}{2}$ -----	. 020	. 010	102	331	115	63
$1\frac{3}{4}$ -----	. 022	. 010	87	284	98	54
2-----	. 024	. 013	76	248	86	48
$2\frac{1}{4}$ -----	. 026	. 013	68	220	76	42
$2\frac{1}{2}$ -----	. 028	. 013	61	199	69	38
$2\frac{3}{4}$ -----	. 030	. 013	56	181	63	35
3-----	. 032	. 013	51	165	57	32

¹ For holes $\frac{3}{8}$ inch in diameter and larger, it is common practice to drill the hole to be reamed with a drill $\frac{1}{64}$ inch undersize. This theoretically leaves 0.015 inch stock to remove on the diameter. However, the drill will cut oversize from 0.001 to 0.005 inch, depending on how accurately it is sharpened.

TABLE VIII.—*Reamer speeds*—Continued

SUGGESTED SPEEDS FOR REAMERS

Material reamed	Suggested speeds, in feet per minute	
	High-speed steel reamers	Carbon-steel reamers
Mild machinery steel 0.2 to 0.3 carbon.....	50 to 70.....	25 to 35.
Steel 0.4 to 0.5 carbon.....	50 to 60.....	25 to 30.
Tool steel 1.2 carbon.....	30 to 40.....	15 to 20.
Steel forgings.....	30 to 40.....	15 to 20.
Alloy steel.....	30 to 50.....	15 to 25.
Stainless steel.....	20 to 30.....	10 to 15.
Soft cast iron.....	70 to 100.....	35 to 50.
Hard chilled cast iron.....	50 to 70.....	25 to 35.
Malleable iron.....	50 to 60.....	25 to 30.
Ordinary brass and bronze.....	130 to 200.....	70 to 100.
High tensile bronze.....	50 to 100.....	25 to 50.
Monel metal.....	20 to 30.....	10 to 15.
Aluminum and its alloys.....	130 to 200.....	70 to 100.
Magnesium and its alloys.....	170 to 300.....	80 to 150.
Bakelite.....	70 to 100.....	35 to 50.

TABLE IX.—*Lathe cutting speeds and lubricants*

Material	Cutting speed (feet per minute) ¹					Lubricant ¹			
	Turning and boring		Threading	Drilling	Reaming	Turning and boring		Threading	Drilling
	Roughing	Finishing				Roughing	Finishing		
Cast iron (soft and medium).	40 to 60	25 to 40	30 to 35	60	20	Dry	Dry	Dry	Dry
Cast iron (hard)	25 to 40	10 to 30	10 to 30	25	20	Dry or a coolant.	Dry or turpentine.	Dry or turpentine or kerosene.	Kerosene.
Soft steel	60 to 120	40 to 75	35 to 50	90	20	Any coolant.	Cutting comp. cutting oil soap water.	Cutting comp. cutting oil soap water.	Any coolant.
Hard steel	20 to 35	10 to 25	10 to 15	35	10	Any coolant.	Mineral lard oil.	Mineral lard oil.	Kerosene strong soda water.
Brass, yellow	150 to 200	100 to 150	Great enough to avoid chatter.	200 to 300	40 to 50	Dry or a coolant.	Dry	Kerosene or turpentine.	Dry
Brass, composition	125 to 150	75 to 100	50 to 75	125 to 200	35 to 55	Dry or a coolant.	Dry, kerosene or turpentine.	Kerosene or turpentine.	Kerosene or turpentine.
Phosphor and manganese bronze.	30 to 80	25 to 60	20 to 40	50	20	Any coolant.	Mineral lard oil.	Mineral lard oil.	Dry or mineral lard oil.
Monel metal	50 to 60	25 to 35	20 to 35	50 to 60	15 to 20	Dry or any coolant.	Mineral lard oil.	Mineral lard oil.	Mineral lard oil.
Aluminum	125 to 150	80 to 125	Great enough to avoid chatter.	125 to 150	35	Dry or kerosene.	Kerosene	Kerosene	Dry
									Kerosene with 25 percent soluble cutting oil.

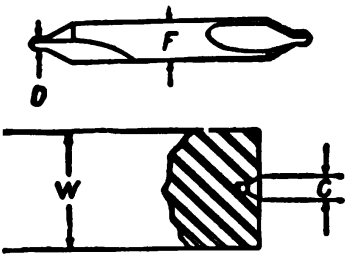
THE MACHINIST

Copper-----	60 to 80----	50 to 60----	30 to 40----	80-----	20-----	Dry or lard oil and turpen- tine.	Mixture of lard oil and tur- pentine.	Dry or a mixture of lard oil and turpentine.	Dry, cooling compound, lard oil, and turpentine.	Dry.
Babbitt-----	100 to 150----	75 to 125----	Great enough to avoid chatter.	150-----	40 to 50----	Dry-----	Mixture of kero- sene and min- eral lard oil.	Mixture of kero- sene and min- eral lard oil.	Kerosene or tur- pentine.	Kerosene

¹ Cutting speeds are approximate only. The proper lubricant to be used must often be determined by experiment.

QUARTERMASTER CORPS

TABLE X.—Size of center hole for 3/16-inch to 4-inch diameter shafts



Diameter of work (W) (inch)	Large diam- eter of coun- tersunk hole (C) (inch)	Diameter of drill (D) (inch)	Diameter of body (F) (inch)
3/16 to 5/16-----	1/8	1/16	1/4
3/8 to 1-----	3/16	3/32	3/10
1 1/4 to 2-----	1/4	1/8	3/10
2 1/4 to 4-----	5/16	5/32	7/16

TABLE XI.—Cutting speeds for turning

The most efficient cutting speed for turning varies with the kind of metal being machined, the depth of the cut, the feed and the type of cutter bit used. If too slow a cutting speed is used, much time may be lost and if too high a speed is used, the tool will dull quickly. The following cutting speeds are recommended for high-speed steel cutter bits:

CUTTING SPEEDS IN SURFACE FEET PER MINUTE

Kind of metal	Roughing cuts 0.010 inch to 0.020 inch feed (feet per minute)	Finishing cuts 0.002 inch to 0.010 inch feed (feet per minute)	Cutting screw threads (feet per minute)
Cast iron-----	60	80	25
Machine steel-----	90	100	35
Tool steel, annealed-----	50	75	20
Brass-----	150	200	50
Aluminum-----	200	300	50
Bronze-----	90	100	25

If a cutting lubricant is used, the above speeds may be increased 25 percent to 50 percent. When using tungsten-carbide tipped cutting tools, the cutting speeds may be increased from 100 percent to 800 percent.

To find the number of revolutions per minute required for a given cutting speed, in feet per minute, multiply the given cutting speed by 12 and divide the product by the circumference (in inches) of turned part.

THE MACHINIST

Example: Find the number of revolutions per minute for 1 inch shaft for a cutting speed of 90 feet per minute.

$$\frac{90 \times 12}{3.1416 \times 1} = 343.77 \text{ rpm}$$

TABLE XII.—*Spindle speeds in rpm for turning and boring*

Calculated for average cuts with high-speed steel cutter bits

Spindle speeds for various diameters and metals are listed in the tabulation below to eliminate the necessity of making calculations.

Diameter in inches	Alloy steels 50 feet per minute	Cast iron 75 feet per minute	Machine steel 100 feet per minute	Hard brass 150 feet per minute	Soft brass 200 feet per minute	Aluminum 300 feet per minute
1.....	191	287	382	573	764	1, 146
2.....	95	143	191	287	382	573
3.....	64	95	127	191	254	381
4.....	48	72	95	143	190	285
5.....	38	57	76	115	152	228
6.....	32	48	64	95	128	192
7.....	27	41	55	82	110	165
8.....	24	36	48	72	96	144
9.....	21	32	42	64	84	126
10.....	19	29	38	57	76	114
11.....	17	26	35	52	70	105
12.....	16	24	32	48	64	96
13.....	15	22	29	44	58	87
14.....	14	20	27	41	54	81
15.....	13	19	25	38	50	75
16.....	12	18	24	36	48	72

TABLE XIII.—*Practical sizes of chucks for lathes*

Size of lathe (inch)	Four-jaw independent lathe chuck (inch)	Three-jaw universal geared scroll chuck (inch)
9.....	6	5
10.....	6	5
13.....	7½	6
14½.....	9	7½
16.....	10	7½

TABLE XIV.—*Rules for figuring tapers*

Given	To find	Rules
The taper per foot-----	The taper per inch---	Divide the taper per foot by 12.
The taper per inch-----	The taper per foot---	Multiply the taper per inch by 12.
End diameters and length of taper in inches.	The taper per foot---	Subtract small diameter from large; divide by length of taper, and multiply the quotient by 12.
Large diameter and length of taper in inches, and taper per foot.	Diameter at small end in inches.	Divide taper per foot by 12; multiply by length of taper, and subtract result from large diameter.
Small diameter and length of taper in inches, and taper per foot.	Diameter at large end in inches.	Divide taper per foot by 12; multiply by length of taper, and add result to small diameter.
The taper per foot and two diameters in inches.	Distance between two given diameters in inches.	Subtract small diameter from large; divide remainder by taper per foot, and multiply the quotient by 12.
The taper per foot-----	Amount of taper in a certain length given in inches.	Divide taper per foot by 12; multiply by given length of tapered part.

THE MACHINIST

TABLE XV.—*American national standard screw thread pitches and recommended tap drill sizes*

American national coarse standard thread (N. C.) (formerly U. S. standard)					American national fine standard thread (N. F.) (formerly SAE thread)				
Sizes	Threads per inch	Outside diameter of screw	Tap drill sizes	Decimal equiva- lent of drill	Sizes	Threads per inch	Outside diameter of screw	Tap drill sizes	Decimal equiva- lent of drill
1----	64	0. 073	53	0. 0595	0--	80	0. 060	$\frac{3}{64}$	0. 0469
2----	56	. 086	50	. 0700	1--	72	. 073	53	. 0595
3----	48	. 099	47	. 0785	2--	64	. 086	50	. 0700
					3--	56	. 099	45	. 0820
4----	40	. 112	43	. 0890	4--	48	. 112	42	. 0935
5----	40	. 125	38	. 1015	5--	44	. 125	37	. 1040
6----	32	. 138	36	. 1065	6--	40	. 138	33	. 1130
8----	32	. 164	29	. 1360	8--	36	. 164	29	. 1360
10----	24	. 190	25	. 1495	10--	32	. 190	21	. 1590
12----	24	. 216	16	. 1770	12--	28	. 216	14	. 1820
$\frac{1}{4}$ ----	20	. 250	7	. 2010	$\frac{1}{4}$ ----	28	. 250	3	. 2130
$\frac{5}{16}$ ----	18	. 3125	F	. 2570	$\frac{5}{16}$ ----	24	. 3125	I	. 2720
$\frac{3}{8}$ ----	16	. 375	$\frac{5}{16}$	. 3125	$\frac{3}{8}$ ----	24	. 375	Q	. 3320
$\frac{7}{16}$ ----	14	. 4375	U	. 3680	$\frac{7}{16}$ ----	20	. 4375	$\frac{25}{64}$	. 3906
$\frac{1}{2}$ ----	13	. 500	$\frac{27}{64}$	. 4219	$\frac{1}{2}$ ----	20	. 500	$\frac{29}{64}$	. 4531
$\frac{9}{16}$ ----	12	. 5625	$\frac{31}{64}$	. 4843	$\frac{9}{16}$ ----	18	. 5625	0. 5062	. 5062
$\frac{5}{8}$ ----	11	. 625	$\frac{17}{32}$	. 5312	$\frac{5}{8}$ ----	18	. 625	0. 5687	. 5687
$\frac{3}{4}$ ----	10	. 750	$\frac{21}{32}$	. 6562	$\frac{3}{4}$ ----	16	. 750	$\frac{11}{16}$	. 6875
$\frac{7}{8}$ ----	9	. 875	$\frac{49}{64}$	. 7656	$\frac{7}{8}$ ----	14	. 875	0. 8020	. 8020
1----	8	1. 000	$\frac{7}{8}$	. 875	1----	14	1. 000	0. 9274	. 9274
$1\frac{1}{8}$ ----	7	1. 125	$\frac{63}{64}$	. 9843	$1\frac{1}{8}$ ----	12	1. 125	$\frac{13}{64}$	1. 0468
$1\frac{1}{4}$ ----	7	1. 250	$1\frac{1}{64}$	1. 1093	$1\frac{1}{4}$ ----	12	1. 250	$1\frac{11}{64}$	1. 1718

QUARTERMASTER CORPS

TABLE XVI.—*American national special screw thread pitches (N. S.) and recommended tap drill sizes*

Sizes	Threads per inch	Outside diameter of screw	Tap drill sizes	Decimal equivalent of drill	Sizes	Threads per inch	Outside diameter of screw	Tap drill sizes	Decimal equivalent of drill
$\frac{1}{4}$ ----	$\left\{ \begin{array}{c} 24 \\ 27 \\ 32 \end{array} \right\}$	0. 250	$\left\{ \begin{array}{c} 4 \\ 3 \\ \frac{1}{2} \end{array} \right\}$	$\left\{ \begin{array}{c} 0. 2090 \\ . 2130 \\ . 2187 \end{array} \right\}$	$\frac{1}{8}$ ---	27	. 5625	$\frac{17}{32}$	. 5312
$\frac{3}{8}$ ----	$\left\{ \begin{array}{c} 20 \\ 27 \\ 32 \end{array} \right\}$	. 3125	$\left\{ \begin{array}{c} \frac{1}{4} \\ J \\ \frac{1}{2} \end{array} \right\}$	$\left\{ \begin{array}{c} . 2656 \\ . 2770 \\ . 2812 \end{array} \right\}$	$\frac{5}{8}$ ----	$\left\{ \begin{array}{c} 12 \\ 27 \end{array} \right\}$	. 625	$\left\{ \begin{array}{c} \frac{35}{64} \\ \frac{19}{32} \end{array} \right\}$	$\left\{ \begin{array}{c} . 5469 \\ . 5937 \end{array} \right\}$
$\frac{1}{2}$ ----	$\left\{ \begin{array}{c} 20 \\ 27 \end{array} \right\}$	. 375	$\left\{ \begin{array}{c} \frac{3}{4} \\ R \end{array} \right\}$	$\left\{ \begin{array}{c} . 3281 \\ . 3390 \end{array} \right\}$	$\frac{3}{4}$ ----	$\left\{ \begin{array}{c} 12 \\ 27 \end{array} \right\}$	. 750	$\left\{ \begin{array}{c} \frac{41}{64} \\ \frac{23}{32} \end{array} \right\}$	$\left\{ \begin{array}{c} . 6719 \\ . 7187 \end{array} \right\}$
$\frac{5}{8}$ ----	$\left\{ \begin{array}{c} 20 \\ 27 \end{array} \right\}$	. 4375	$\left\{ \begin{array}{c} \frac{1}{2} \\ X \\ Y \end{array} \right\}$	$\left\{ \begin{array}{c} . 3970 \\ . 4040 \end{array} \right\}$	$\frac{7}{8}$ ----	$\left\{ \begin{array}{c} 12 \\ 18 \\ 27 \end{array} \right\}$	. 875	$\left\{ \begin{array}{c} \frac{51}{64} \\ \frac{53}{64} \\ \frac{27}{32} \end{array} \right\}$	$\left\{ \begin{array}{c} . 7969 \\ . 8281 \\ . 8437 \end{array} \right\}$
$\frac{3}{4}$ ----	$\left\{ \begin{array}{c} 24 \\ 27 \end{array} \right\}$	. 4375	$\left\{ \begin{array}{c} X \\ Y \end{array} \right\}$	$\left\{ \begin{array}{c} . 3970 \\ . 4040 \end{array} \right\}$	1-----	$\left\{ \begin{array}{c} 12 \\ 27 \end{array} \right\}$	1. 000	$\left\{ \begin{array}{c} \frac{59}{64} \\ \frac{31}{32} \end{array} \right\}$	$\left\{ \begin{array}{c} . 9219 \\ . 9687 \end{array} \right\}$
$\frac{1}{2}$ ----	$\left\{ \begin{array}{c} 12 \\ 24 \\ 27 \end{array} \right\}$	0. 500	$\left\{ \begin{array}{c} \frac{27}{64} \\ \frac{29}{64} \\ \frac{15}{32} \end{array} \right\}$	$\left\{ \begin{array}{c} 0. 4219 \\ . 4531 \\ . 4687 \end{array} \right\}$					

TABLE XVII.—*Thread-cutting data*

This table is based on .005" per cut allowing an extra cut for finish.

Number of threads per inch	Number of chasing cuts
8	18
10	14
11	13
12	11
13	10
16	9
20	8

THE MACHINIST

TABLE XVIII.—*Milling machine cutting speeds and lubricants*

Material	Cutting speed (feet per minute) ¹				Lubricant ¹	
	Spiral cutters		Face mills			
	Roughing	Finishing	Roughing	Finishing	Roughing	Finishing
Cast iron (soft and medium).	40 to 75.....	25 to 80.....	35 to 65.....	30 to 80.....	Dry.....	Dry.
Cast iron (hard).	25 to 40.....	10 to 30.....	25 to 40.....	20 to 45.....	Dry or a coolant.	Dry or turpentine.
Soft steel.....	60 to 120.....	45 to 110.....	50 to 85.....	45 to 100.....	Any coolant...	Cutting compound, cutting oil, soap water.
Hard steel.....	25 to 50.....	25 to 70.....	25 to 50.....	25 to 70.....	Any coolant...	Mineral lard oil.
Brass, yellow...	150 to 200...	100 to 250...	100 to 200...	100 to 200...	Dry or a coolant.	Dry.
Brass, composition.	125 to 200...	90 to 200.....	90 to 150.....	90 to 150.....	Dry or a coolant.	Dry kerosene or turpentine.
Phosphor and manganese bronze.	30 to 80.....	25 to 100.....	30 to 80.....	30 to 80.....	Any coolant ..	Mineral lard oil.
Monel metal....	50 to 75.....	50 to 75.....	40 to 60.....	40 to 60.....	Dry or any coolant.	Mineral lard oil.
Aluminum.....	400 to 1,000..	400 to 1,000..	400 to 1,000..	400 to 1,000..	Dry or kerosene.	Kerosene.

¹ Cutting speeds are approximate only. The proper lubricant to be used must often be determined by experiment. Flood cooling is always advisable when a coolant is used.

(This table represents approved practices with high-speed steel milling cutters. The speed and feed for production milling are often limited by the ability of a machine to withstand the heavy cuts. Run carbon-steel cutters at about half the speeds given above.)

QUARTERMASTER CORPS

TABLE XIX.—*Shaper cutting speeds and feeds*

Materials	Cutting speed (feet per minute) ¹				Lubricant	
	Roughing		Finishing		Roughing	Finishing
	H. S. tools ²	C. S. tools ³	H. S. tools	C. S. tools		
Cast iron (soft and medium).	60	30	45	15	Dry.....	Dry.
Cast iron (hard)	35	20	20	10	do	Strong soda water.
Soft steel	Highest possible	35	50	25	Cutting compound..... Soda water, soap water.	Soda water. Do.
Hard steel	35	15	25	10	Mineral lard oil..... Cutting compound.	Mineral lard oil.
Brass	Highest possible	35	80	30	Dry.....	Dry.
Phosphor bronze....	50	20	30	15	do	Do.
Monel metal	50	20	30	15	Mineral lard oil	Mineral lard oil.
Aluminum	125	50	60	40	Dry.....	Kerosene.
Copper	80	40	30	20	do	Dry.
Babbitt.....	Highest possible	50	80	30	Kerosene, plain or mixed with lard oil.	Kerosene and lard oil.

¹ Cutting speeds are approximate only. In special cases the proper lubricants must be determined by experiment.

² High-speed steel.

³ Carbon-steel.

THE MACHINIST

APPENDIX II

BIBLIOGRAPHY

The following sources have been consulted for illustrations and text material in the preparation of this manual. They contain more detailed information on machine shop practice than is contained herein, and should be consulted as collateral reading.

Barritt, J. W., *Machine Shop Operations* (Chicago, Ill.: American Technical Society, 1939).

Burghardt, Henry D., *Machine Tool Operation* (Part II, New York, N. Y.: McGraw-Hill Book Company, Inc., 1922).

Colvin, Fred H., and Stanley, Frank A., *American Machinists' Handbook* (Seventh Edition, New York, N. Y.: McGraw-Hill Book Company, Inc., 1940).

How To Run a Lathe (Thirty-Ninth Edition, South Bend, Ind.: South Bend Lathe Works, 1940).

Kent's Mechanical Engineers Handbook (Eleventh Edition, Vol. III, Design Shop Practice, New York, N. Y.: John Wiley & Sons, Inc., 1938).

Modern Shop Practice (Vol. I, Revised Edition, Chicago, Ill.: American Technical Society, 1939).

Running a Regal (Fourth Edition, Cincinnati, Ohio: The R. K. LeBlond Machine Tool Company, 1940).

Shop Theory (Revised Edition, Dearborn, Mich.: Henry Ford Trade School, 1941).

Smith, Robert H., *Text-Book of Advanced Machine Work* (Twelfth Edition, Boston, Mass.: Industrial Education Book Company, 1940).

Starrett Books (Vol. I, Twenty-Third Edition, *The Starrett Book For Machinists' Apprentices*; Vol. II, Thirteenth Edition, *The Starrett Book For Machinists*; Vol. III, Eighth Edition, *The Starrett Book*

QUARTERMASTER CORPS

[A. G. 062.11 (8-21-41).]

BY ORDER OF THE SECRETARY OF WAR:

G. C. MARSHALL,
Chief of Staff.

OFFICIAL:

E. S. ADAMS,
Major General,
The Adjutant General.

DISTRIBUTION :

B 1 (6), 2, 4, 6 (2); R 1 (6), 2, 4-7, 17 (5), 10 (15); Bn 5, 10, 17 (5), 9 (2); IBn 1, 2, 6 (5); C 10, 17 (8); IC 2, 4-7, 10, 17 (10), 3 (5).

(For explanation of symbols see FM 21-6.)

